

A decorative graphic on the left side of the slide, consisting of three overlapping circular frames. The top frame shows a worker in a red bucket on a lift. The middle frame shows a large industrial building with a curved roof and zebras in the foreground. The bottom frame shows high-voltage power lines against a blue sky.

# Spherical Header Tutorial

From primitive parts

***Stefan van der Walt***

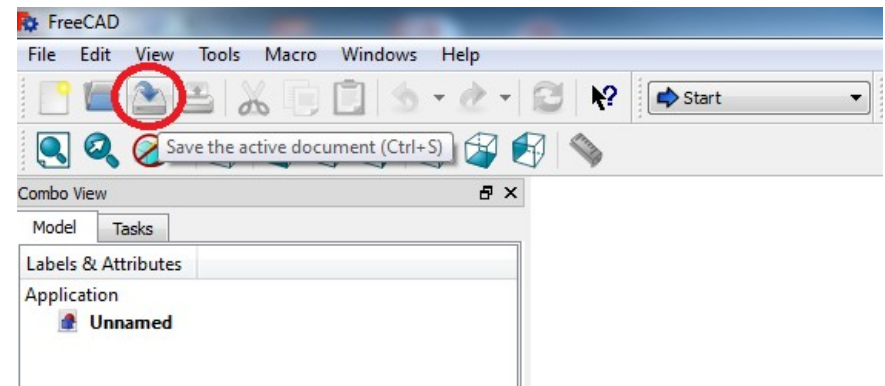
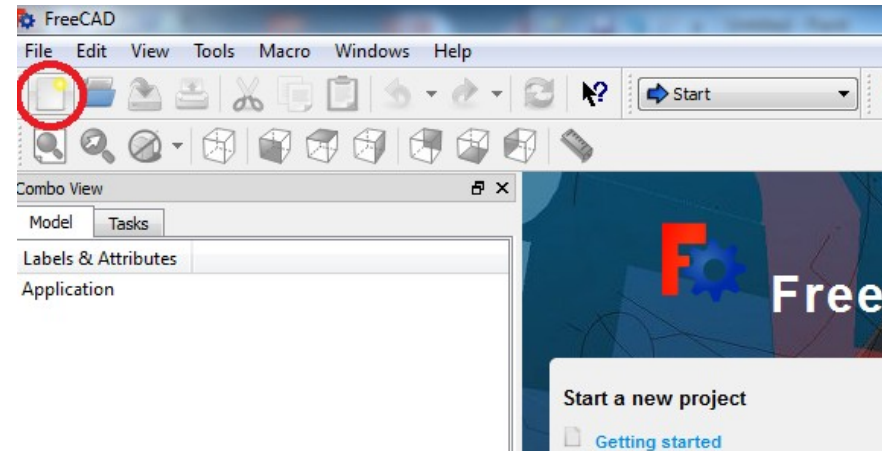
[vdwalts@eskom.co.za](mailto:vdwalts@eskom.co.za)

09/11/2016

- In this tutorial a basic spherical header with two branches will be created using primitives.
- A static finite element analysis will then be set up and solved.
- The results will be displayed with the built-in results view.
- The results will then be displayed using the VTK post-processor.

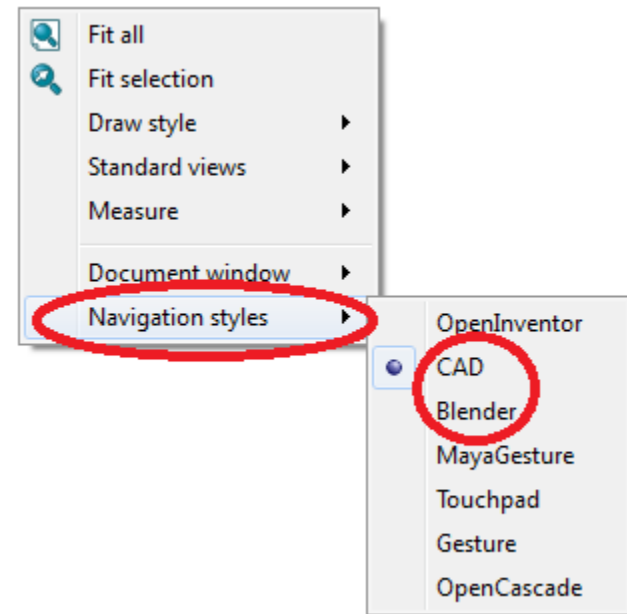
## Create the Project:

- When you open Freecad, the home screen appears.
- Create a new document by clicking on the 'Create a new empty document' icon.
- It is a good idea to save the document before starting. Click on the 'Save' icon on the taskbar. Name the file "Basic Header Tutorial".
- Remember to save regularly.



## Basic navigation











- There are different navigation modes available within FreeCAD. If you right-click anywhere on an open part of the display window, you can select the navigation mode you prefer.
- The 'CAD' and 'Blender' navigation styles are two of the most commonly used styles.








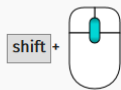


## Rotate parts with navigation styles

- It is important to know how to rotate parts with these navigation styles.
- If the Blender style is selected, a part can be rotated by clicking and holding the middle mouse button.
- If the CAD style is selected, you can rotate a part by clicking and holding both the middle mouse button and the left mouse button.
- The images on the right summarises the main mouse functions.

## CAD Navigation

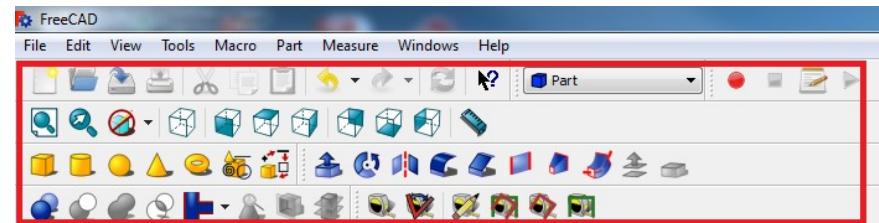
Select	Pan	Zoom	Rotate View	Rotate View Alternate Method
				
				

## Blender Navigation

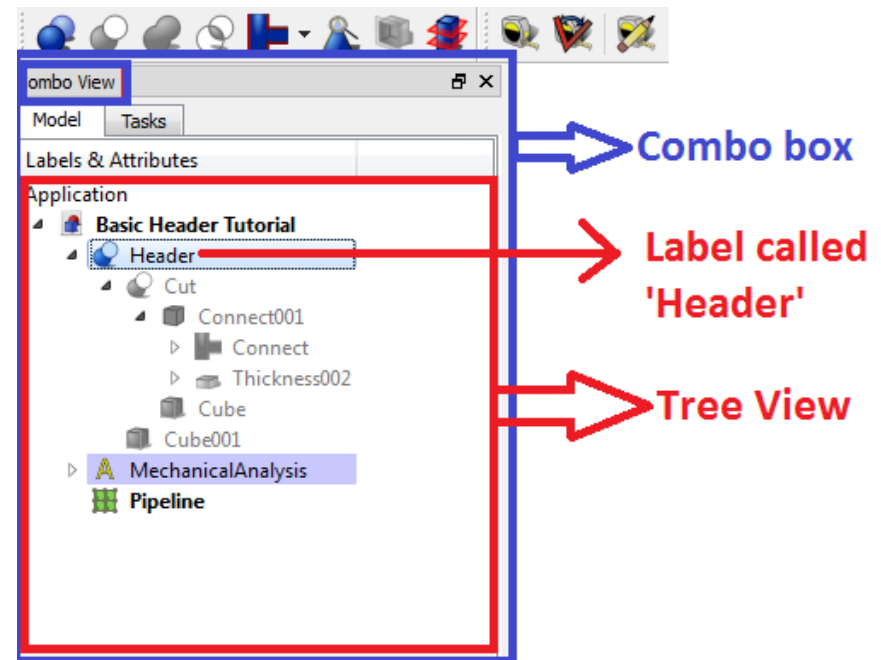
Select	Pan	Zoom	Rotate View
			
			

## Screen layout and terminology

- Certain terms will be used throughout the training material, which will be shown here.
- The taskbar refers to the area at the top of the screen where all the relevant icons are located. The icons will change whenever the workbenches are changed. The example shown here is of the taskbar with the general icons as well as the Part workbench icons.
- The entire area enclosed within the red box is referred to as the taskbar.

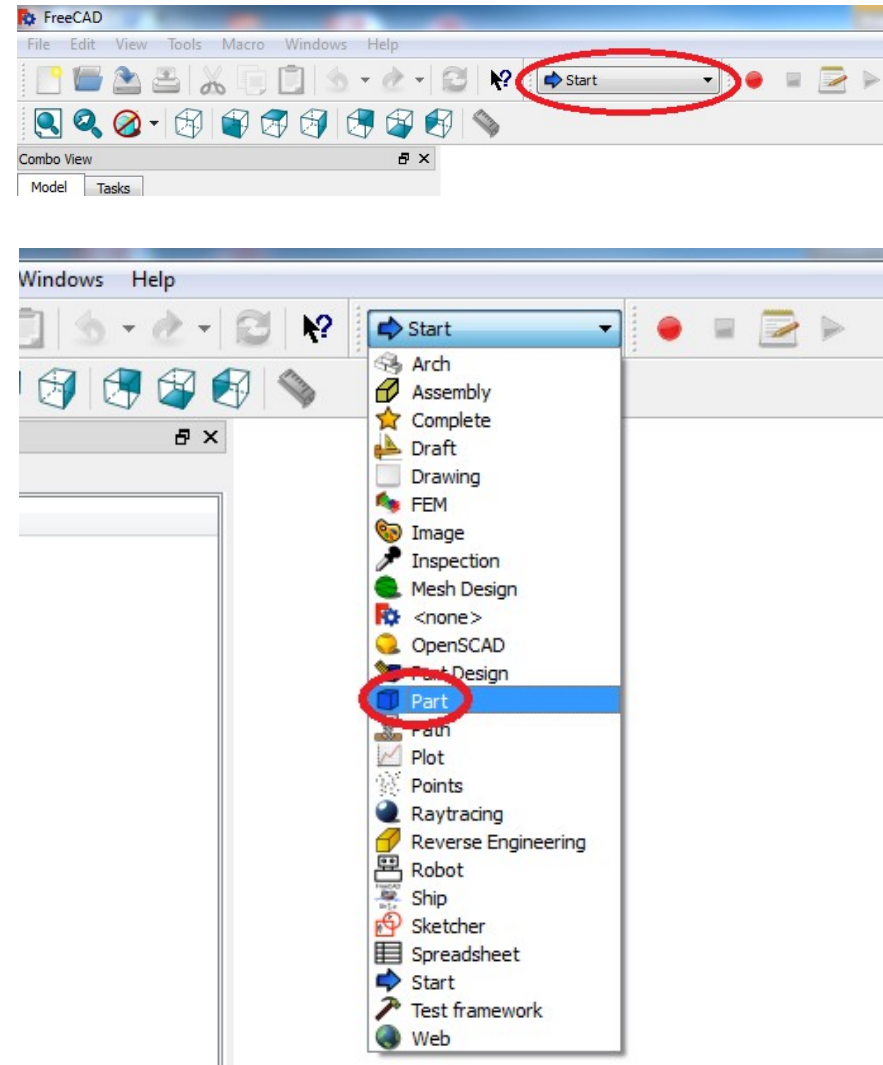


- Other terms often used are the tree view, combo box and label.
- The tree view, which is contained within the combo box, refers to the window in which all the parts and features that was created and applied to the model are displayed.
- In the example shown, the tree view contains features added with the parts workbench as well as features added with the FEM workbench. Each item in the list is called a 'Label' and has a name. If you click on a label you can edit its properties which appear underneath the tree view.
- You can hide a label by clicking on it<sup>7</sup>



## Create the Project:

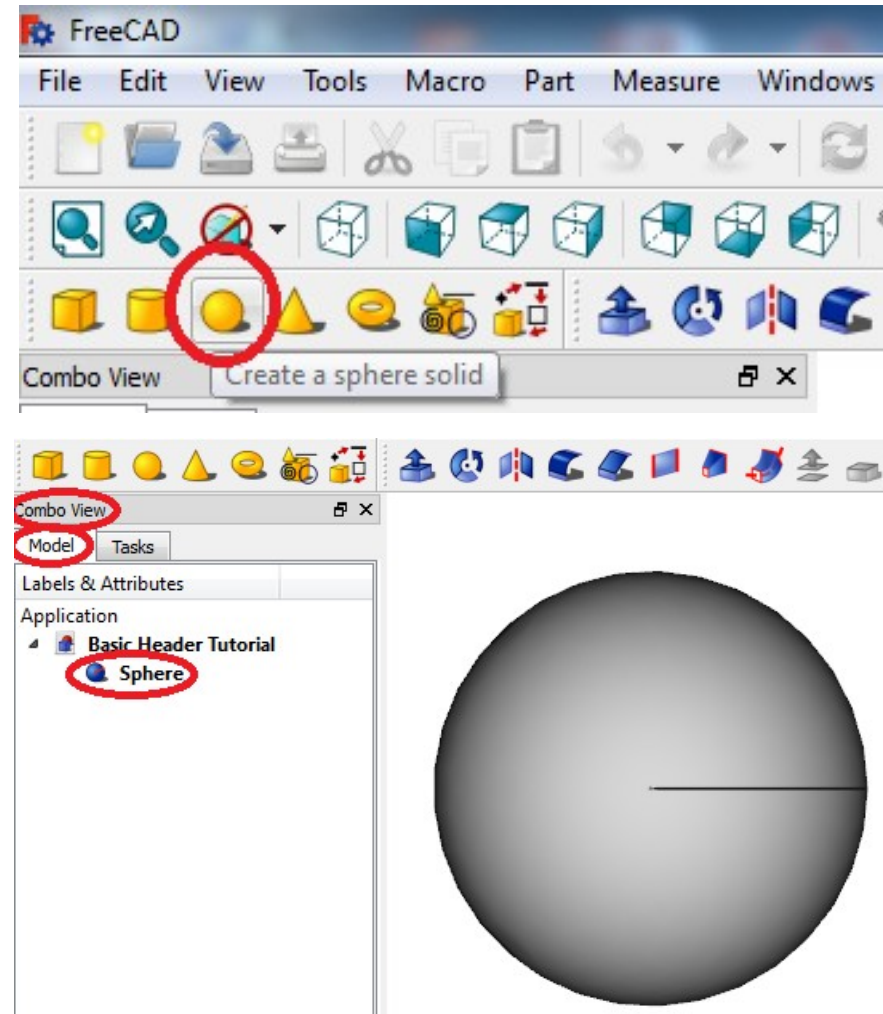
- The workbench dropdown menu is located at the top of the screen on the taskbar. If FreeCAD has just been opened, it will display 'Start'.
- Click on the workbench dropdown menu. The menu will expand and display all the available workbenches within FreeCAD.
- Click on the 'Part' workbench.





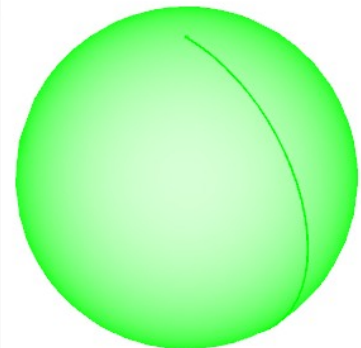
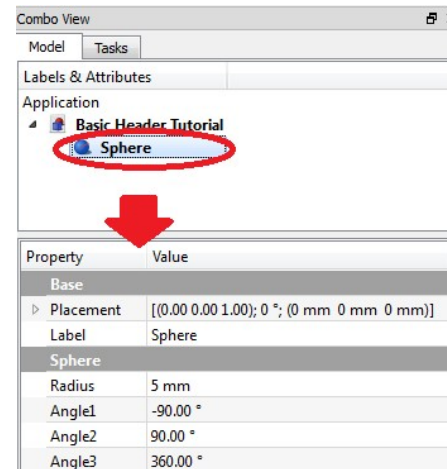
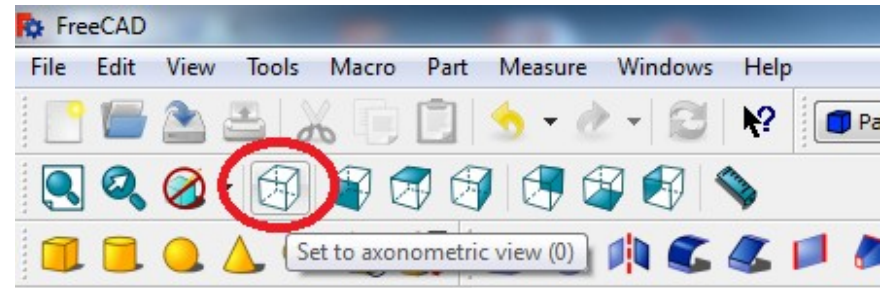
## Spherical Part

- To insert a sphere into your document, click on the 'Sphere' primitive on the taskbar.
- In the Combo View box under the 'Model' tab a sphere will appear in the tree view of the document.



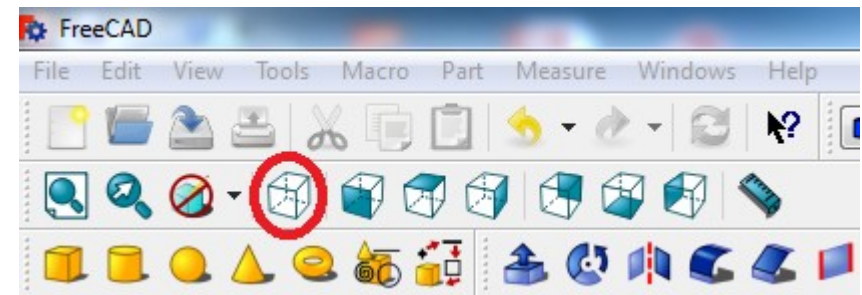
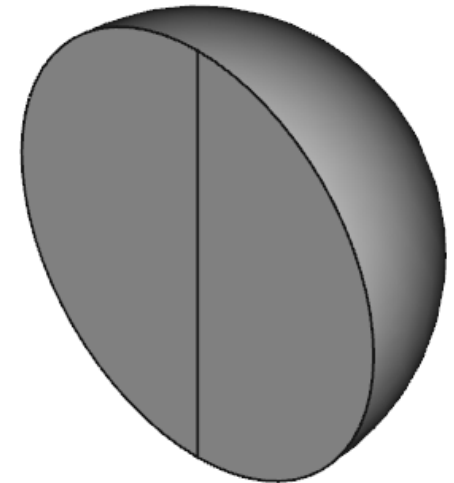
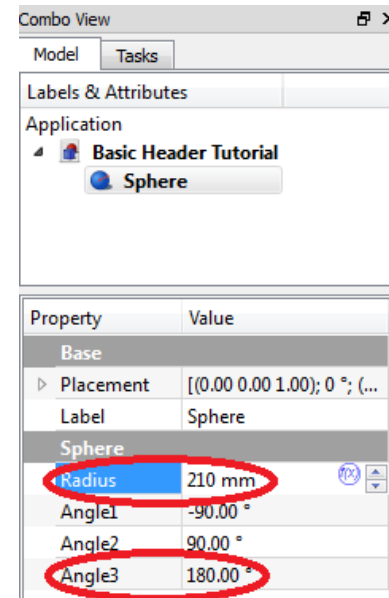
## Halving the sphere

- First we need to set the display window to 'axonometric view'. This can be done by clicking on the icon on the taskbar
- Hint: You can access the shortcuts for the 7 different views by pressing the '0' to '6' keys on your keyboard. 'Axonometric view' is '0'.
- Click on the 'Sphere' label in the tree view to display its properties.




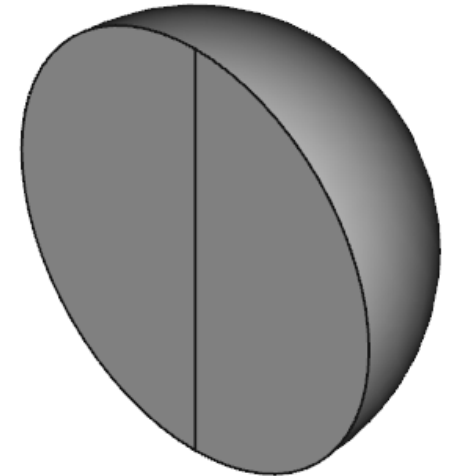
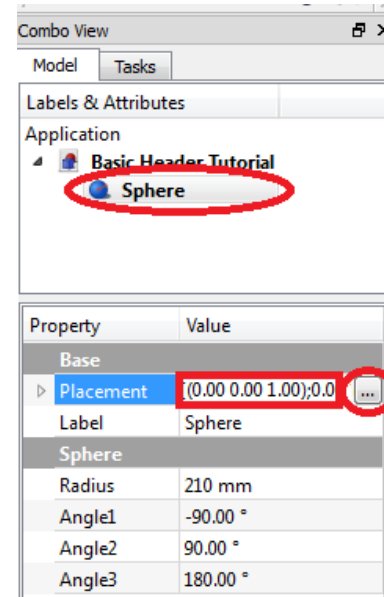
# CAD Model - Halving the sphere

- In the 'Properties' box, change the 'Angle3' to 180°. This will change the sphere to half a sphere. (i.e. if you change it to 90° it will change to a quarter of a sphere etc.).
- In the same properties box, change the radius of the sphere to 210mm.
- Since the original radius was 5mm, the sphere is now too big to be visible on the screen. In order to fit the enlarged sphere in your screen, click on the 'Fit on screen' icon on the taskbar

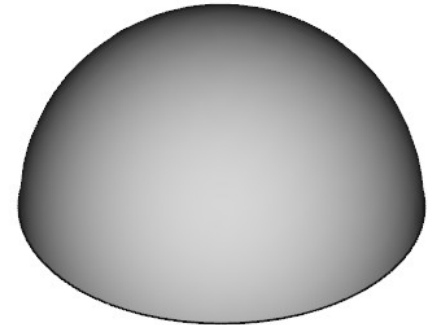
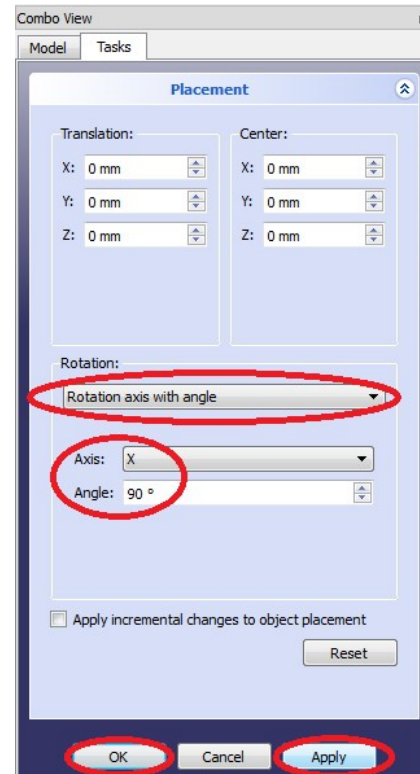


## Rotating the sphere


- It can clearly be seen that the half sphere need to be rotated in order for the 'bottom' part of the half sphere to be downwards.
- In order to rotate the half sphere, select to sphere to open its 'Property' box, and click on the box next to placement. A small icon with an ellipse on it appears . Click on it to open the 'Placement' properties of the half sphere.

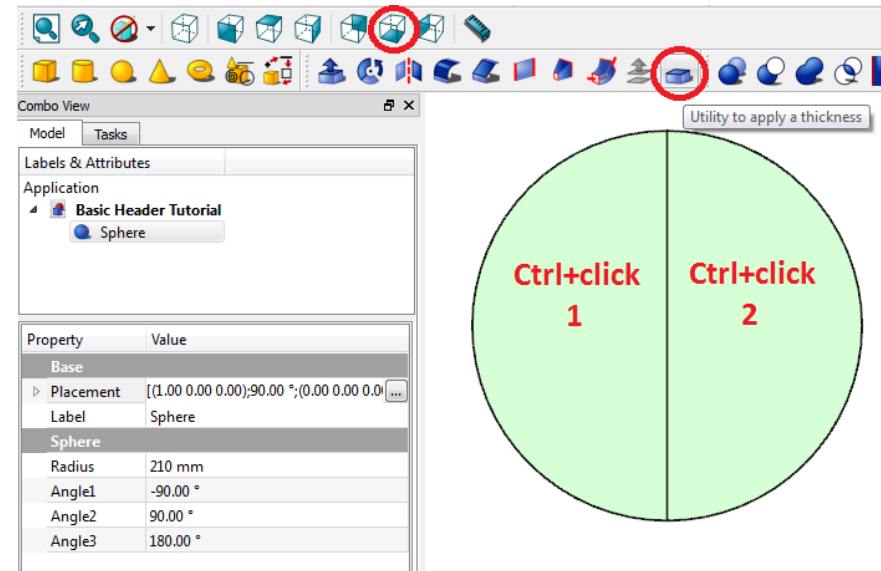


- After opening the 'Placement' properties for the half sphere, we need to select the 'Rotation axis with angle' in the dropdown menu under 'Rotation'.
- Select the 'X' axis next to the 'Axis' heading.
- Change the angle to 90°.
- Click 'Apply' to apply the changes. The half sphere will have rotated with its 'bottom' part downward.
- Click 'OK' to close the box.

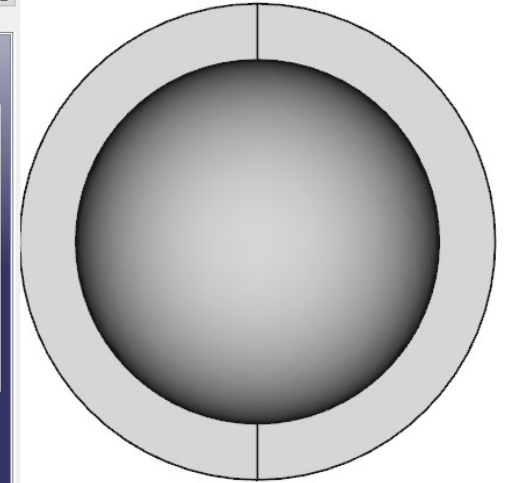
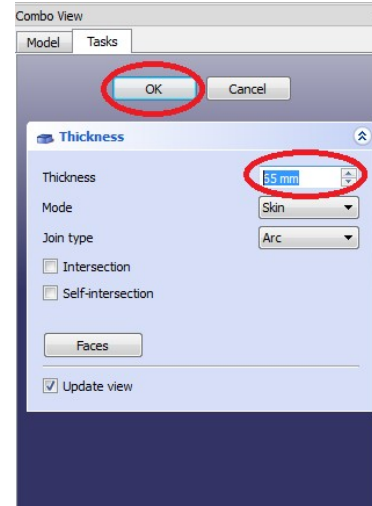


## Applying a thickness to the sphere

- In order to apply a thickness (hollowing out the sphere), we need to rotate the half sphere until the bottom part is visible. This can be done either by selecting the 'Bottom' view  from the taskbar or by pressing its shortcut on the keyboard '5'.
- Now select the bottom surface(s) by pressing and holding the 'Ctrl' button while clicking.
- When all the bottom surfaces have been selected, click on the 'Utility to apply a thickness' button on the taskbar. Its box automatically opens.





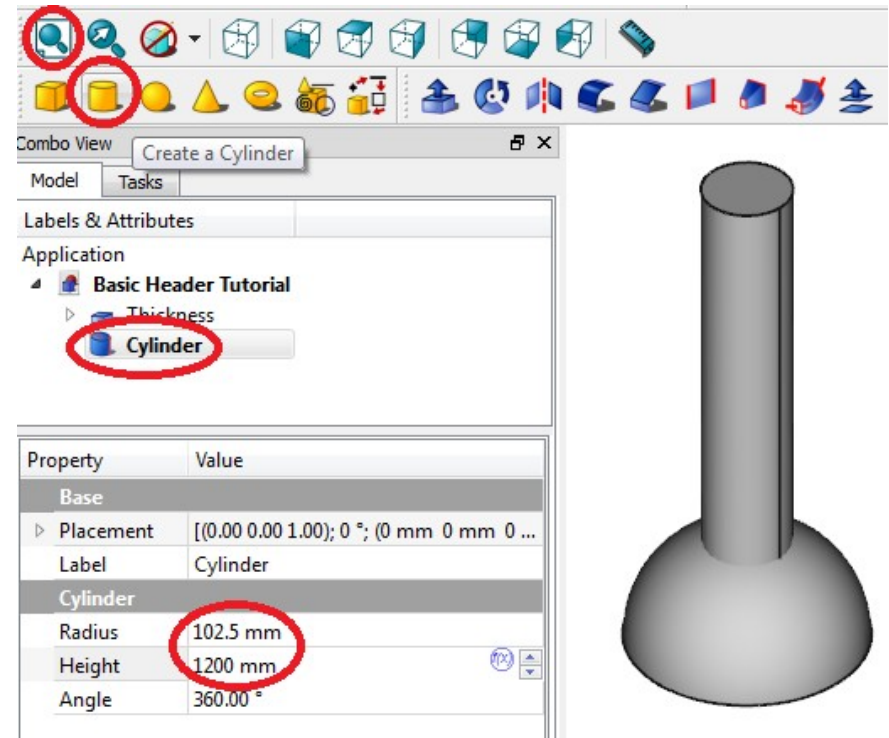
- After selecting the 'Utility to apply a thickness', a box will automatically open.
- Change the thickness to 65mm. You will see the thickness being added to the outside of the sphere.
- Click 'OK' to apply the changes and close the dialogue box.





## Adding a cylinder for the branch

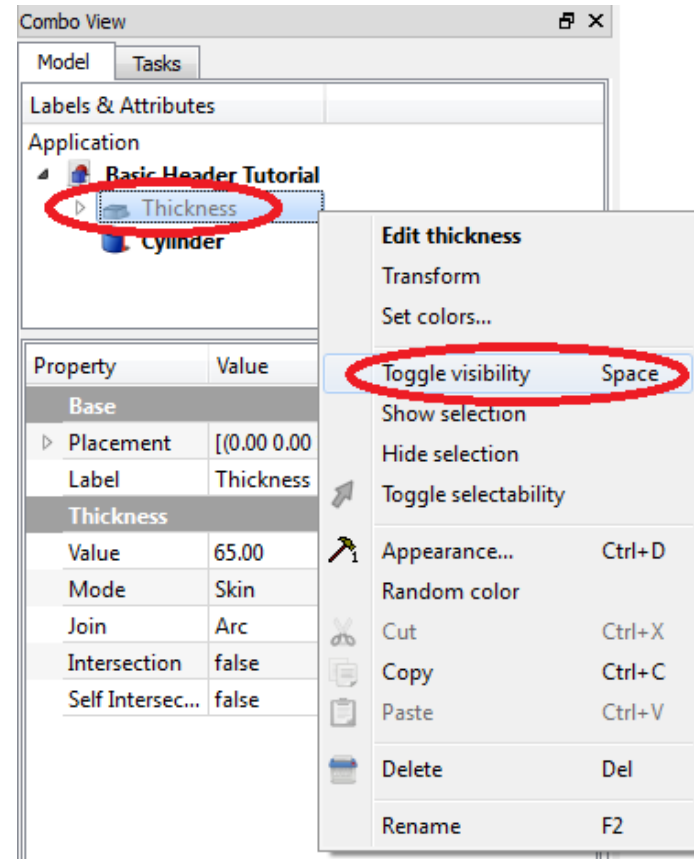
- Select the axonometric view again by selecting its icon  on the taskbar or by pressing the number '0' key on your keyboard.
- Click on the 'cylinder' primitive in the taskbar to insert a cylinder into the document.
- You will see the cylinder label appear in the tree view. Click on it to open its properties box.
- Change its radius to 102.5mm and its length to 1200mm.
- Click  on the 'Fit content to window' icon to zoom out.



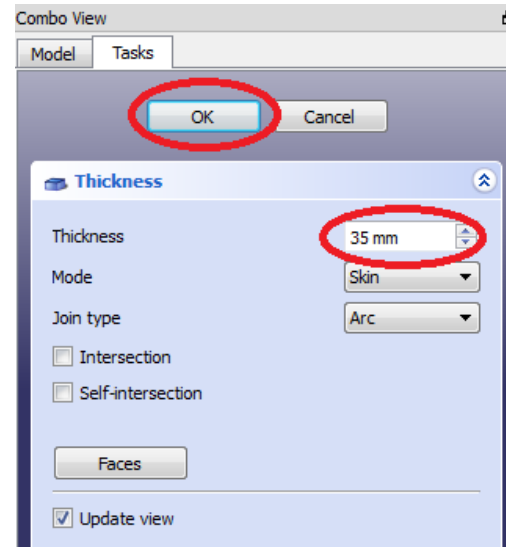


## Applying a thickness to the cylinder

- To apply a thickness to the cylinder in order to change it to a pipe, we need to be able to access both the top and bottom surfaces. But the half sphere might be in the way, so we need to hide the sphere temporarily.
- In order to hide the sphere, either right click on the 'Thickness' label on the tree view and select 'Toggle visibility', or press space bar on our keyboard as a shortcut.
- You will see the half sphere disappear and the 'Thickness' label on the tree view will grey out.

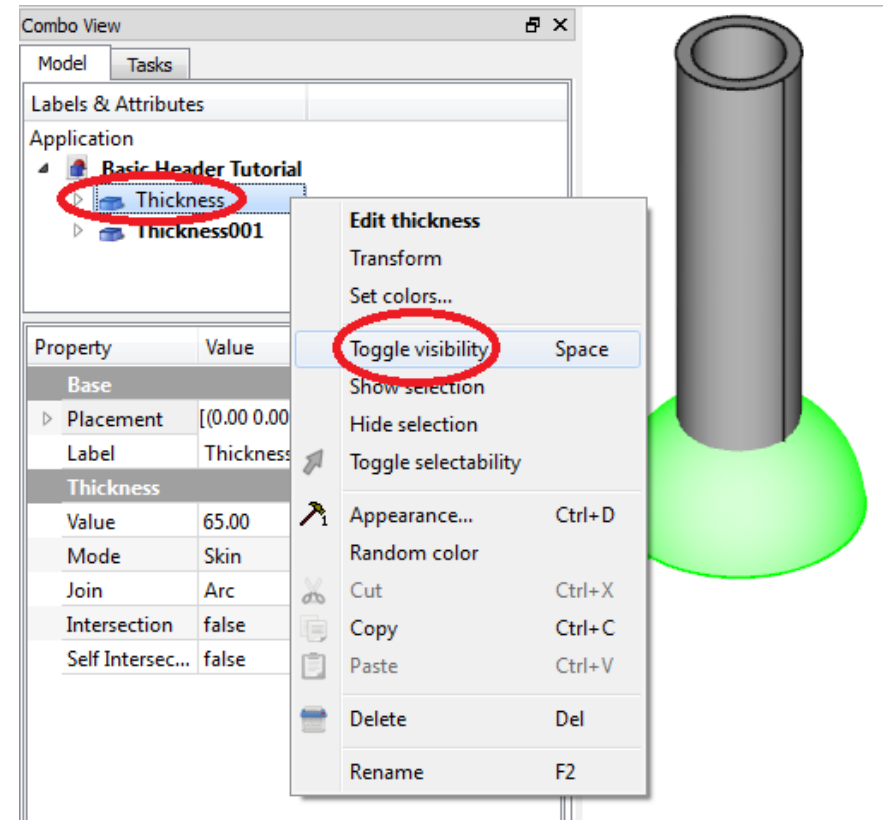


- Select the top and bottom surface of the cylinder by using 'ctrl+click'.
- After selecting both surfaces, click on the 'Utility to apply thickness' in the taskbar.
- Set the thickness to 35mm.
- Click 'OK' to apply the thickness and close the dialogue.

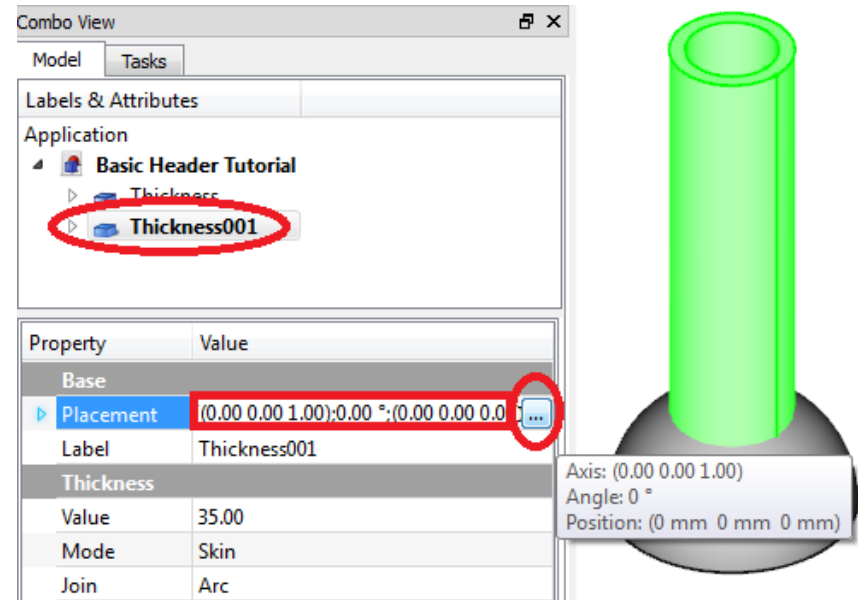


## Rotating the branch

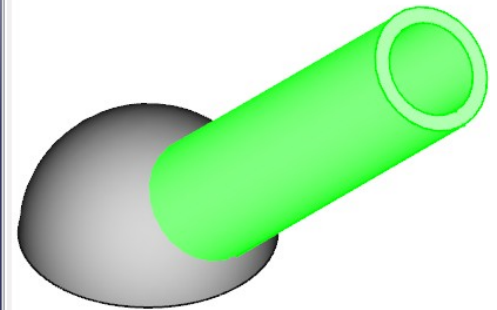
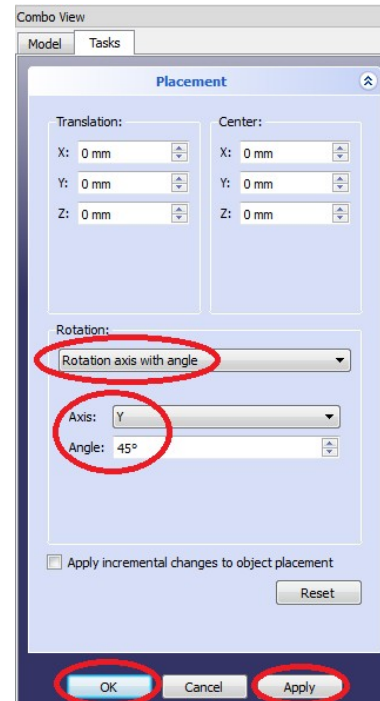
- Since the branch of the spherical header is not upright but at an angle, we need to rotate the branch around the y-axis.
- In order to visualise the relative rotation to the spherical part of the header, we need to toggle the visibility of the half sphere.
- Click on the greyed-out 'Thickness' label in the tree view and press 'spacebar' (or right click on the label and select 'Toggle visibility'.
- Both the spherical and the cylindrical part of the header should be visible.



- Select the label 'Thickness001' which contains the cylindrical branch.
- In its properties box, click on the cell next to 'Placement'. Click on the icon with the ellipse that appear.
- A dialogue box opens.



- Under 'Rotation', select 'Rotation axis with angle'.
- Next to 'Axis', select 'Y' –axis.
- Next to 'Angle', type in '55' to rotate the branch 45°.
- Click on 'Apply' to apply the changes and on 'OK' to close the dialogue box.

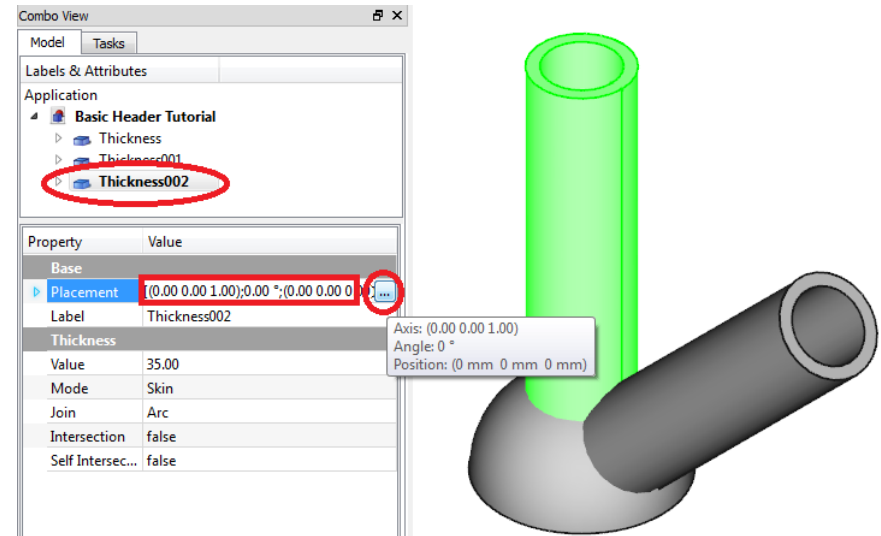


## Adding a second branch

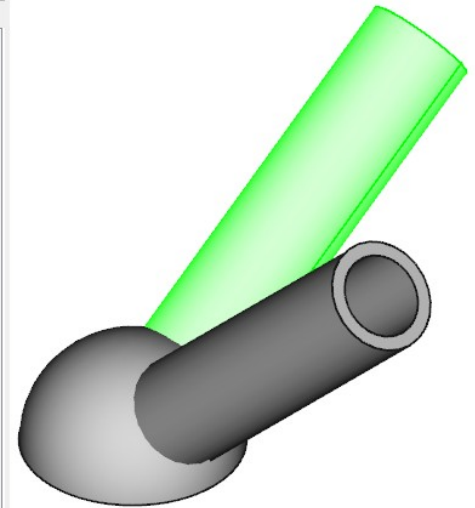
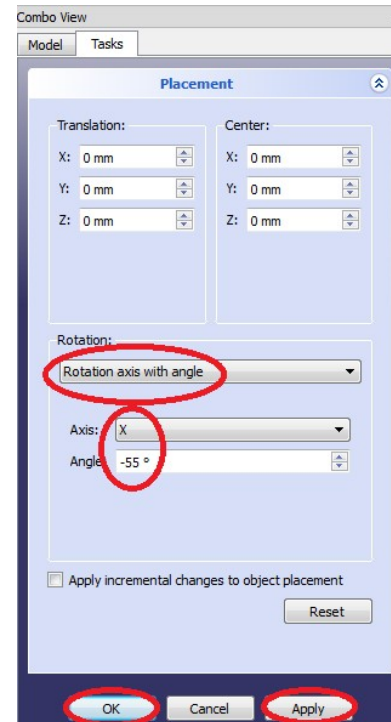
- To add a second branch, follow the exact same steps to add a cylinder.
- Give the cylinder the same dimensions as the first cylinder – make the radius 102.5mm and the length 1200mm.
- Toggle the visibility of both the 'Thickness' labels by clicking on each one individually and pressing spacebar.
- Select the top and bottom surfaces of the cylinder and apply a thickness of 35mm.

## Rotating the second cylinder

- Toggle the visibility of the hidden parts again by using spacebar.
- Click on the cylinder's label in the tree view. It should be 'Thickness002'.
- In the properties box, click on the cell next to placement and click on the icon with ellipse that appear.
- A dialogue box appear.



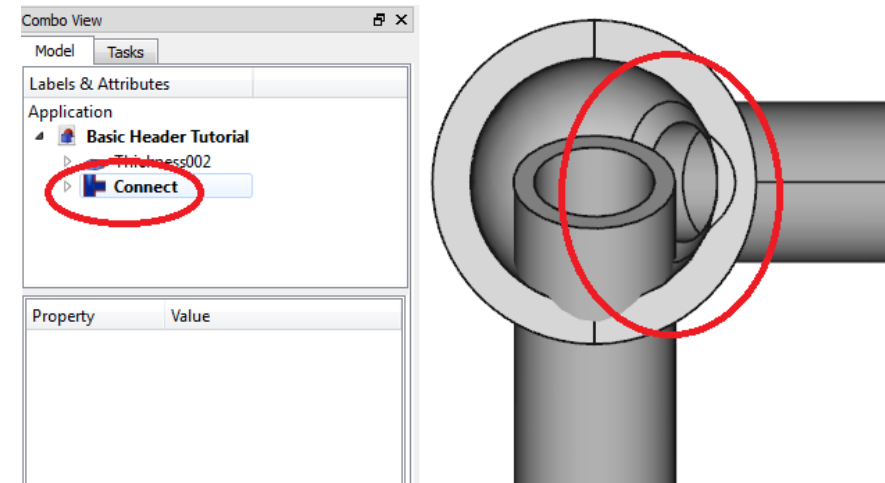
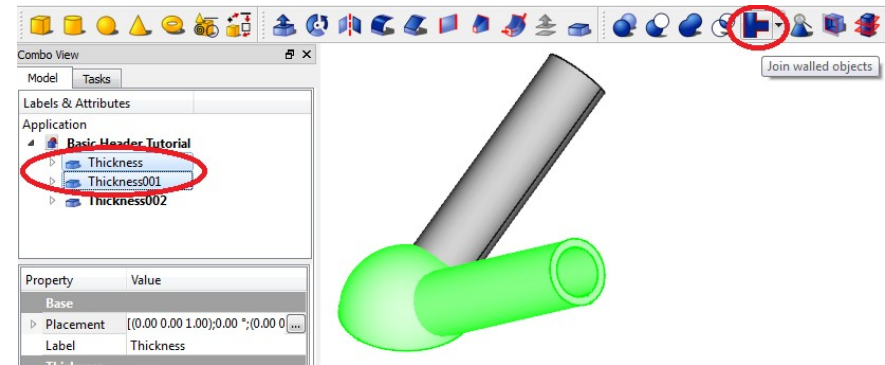
- Under 'Rotation', select 'Rotation axis with angle'.
- Next to 'Axis', select 'X' to rotate the cylinder around the x-axis.
- Next to 'Angle', type in '-55' to rotate the cylinder 55° in the negative (backwards) direction.
- Click on 'Apply' to apply the changes and on 'OK' to close the dialogue.



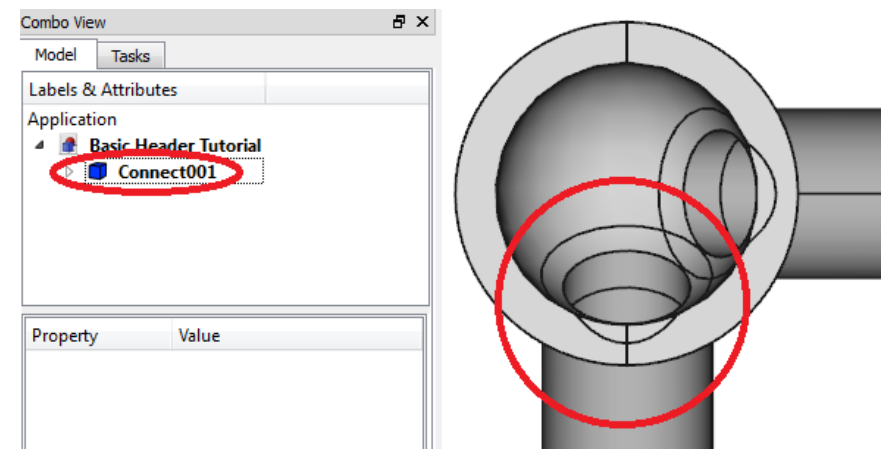
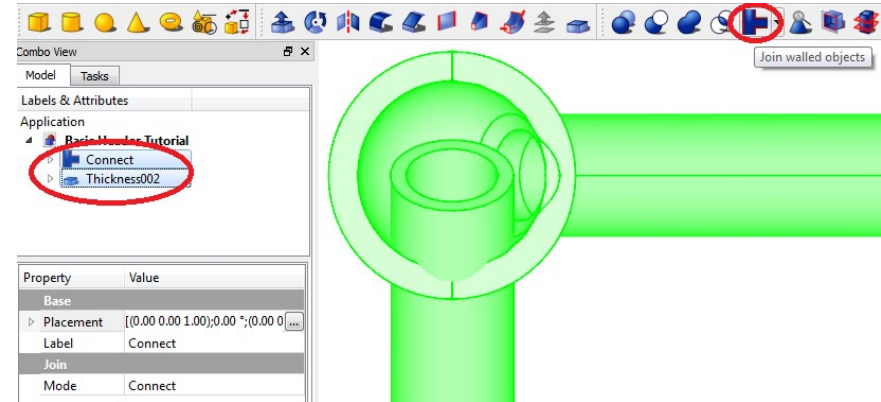


## Join walled objects

- The 3 different parts need to be joined in one single part. This must be done one branch at a time.
- In the tree view, select the first 'Thickness' label and 'ctrl+click' the second label to select both.
- Click on the 'Join walled object' icon in the taskbar.
- This joins the two parts and creates a new label in the tree view called 'Connect'.

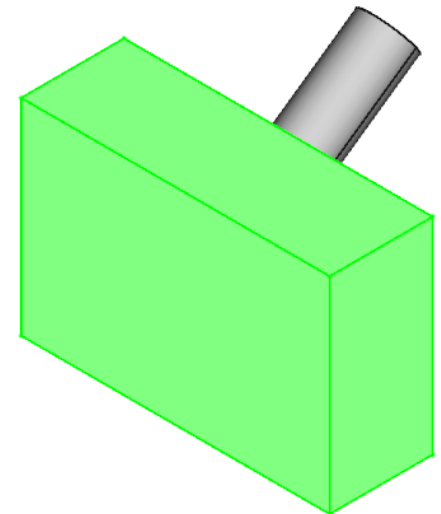
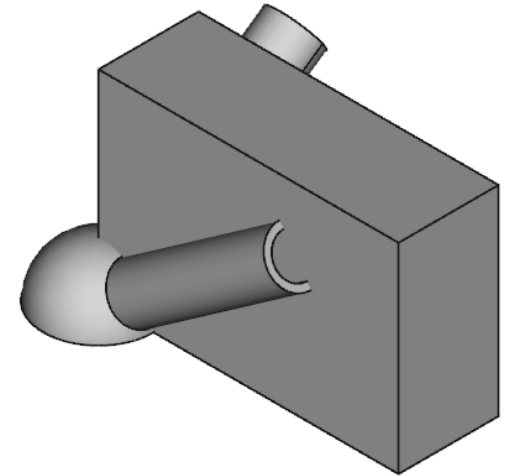
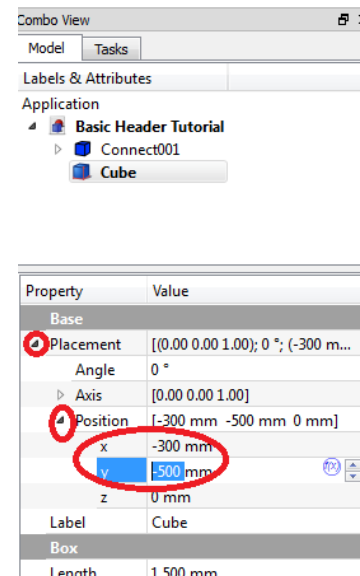
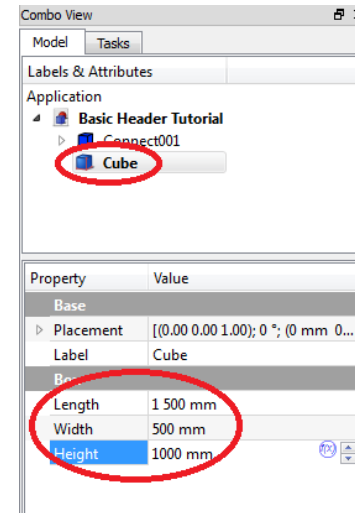


- In order to join the second branch to the body, click on the 'Connect' label in the tree view and then 'ctrl+click' on the remaining 'Thickness002' label.
- Click on the 'Join walled objects' icon in the taskbar.
- A new label 'Connect001' appears and all the other labels are now contained within this label.

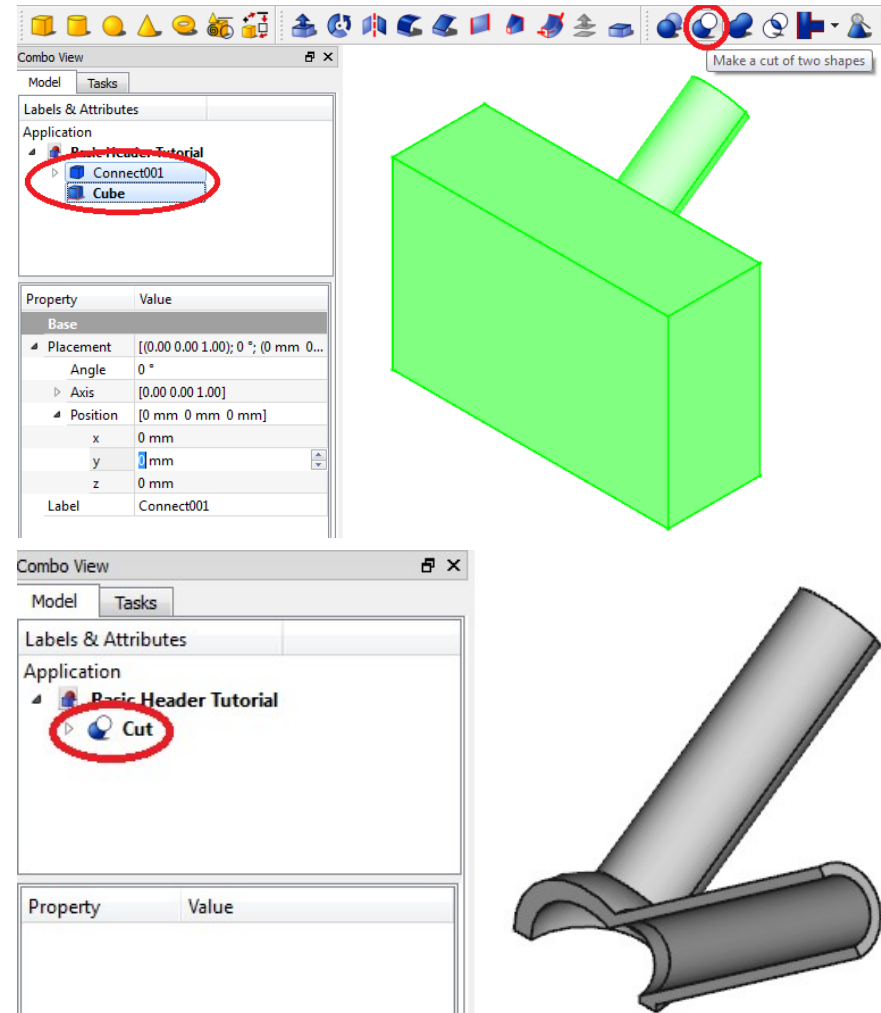


## Cutting the geometry

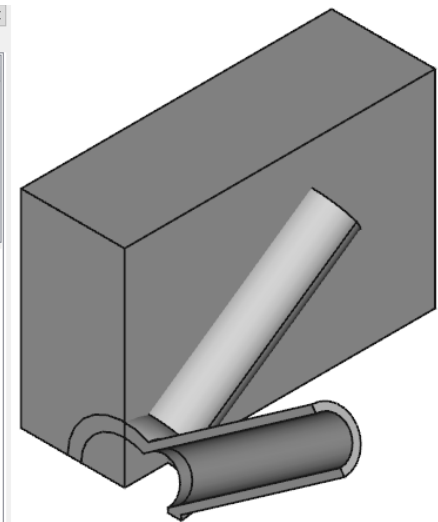
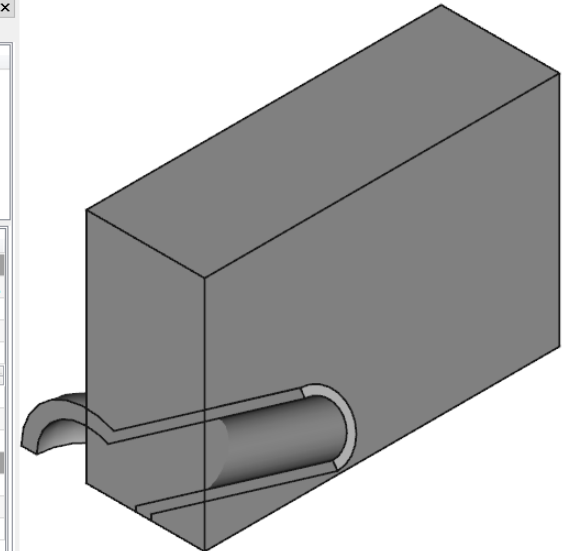
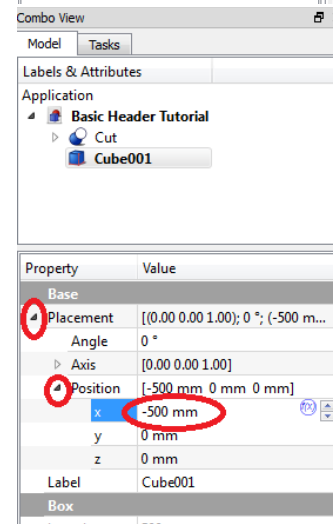
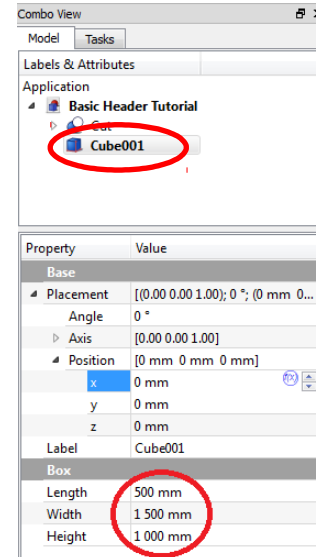
- Due to symmetry, we can cut the part until only a quarter is left.
- In order to do this, click on the 'Cube' icon in the taskbar to insert a cube primitive.
- Click on the 'Cube' label in the tree view and change its length to 1500mm, its width to 500mm and its height to 1000mm.
- Expand the 'Placement' heading by clicking on the white arrow next to it, and then expand 'Position'.
- Change the x and y-coordinate to -300mm and -500mm respectively.



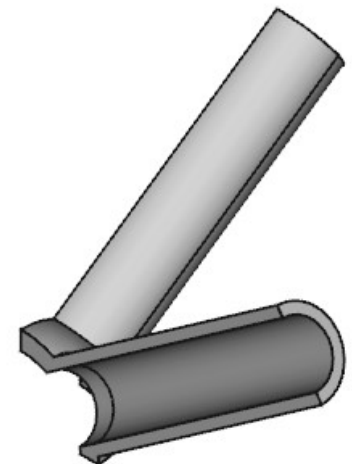
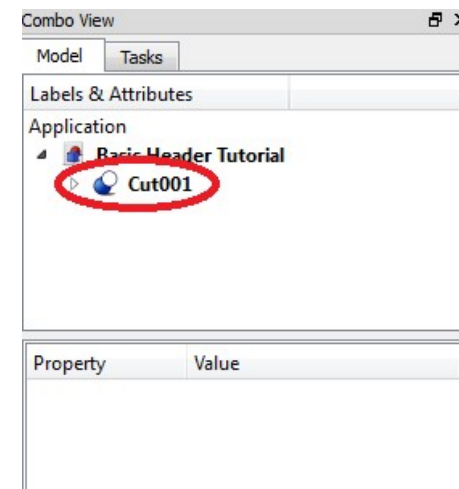
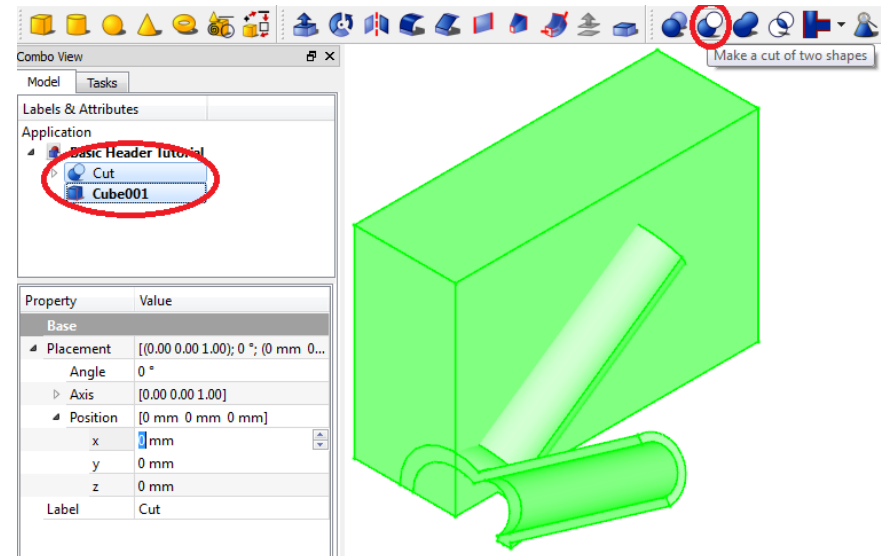
- In order to cut the part, we first need to select the part we want to keep, and after that select the part we want to use to cut away the material.
- In this case, click on the 'Connect' label in the tree view first. Now 'ctrl+click' the 'Cube' label in the tree view.
- Click on the 'Make a cut of two shapes' icon in the taskbar.
- A new label appear in the tree view.



- The part is now cut in half, now we need to cut the remaining part in half as well.
- Click on the 'Cube' icon in the taskbar to insert a cube.
- Click on the 'Cube001' label in the tree view. Under its properties, change its length to 500mm, its width to 1500mm and its height to 1000mm.
- Expand the 'Placement' heading by clicking on the white arrow then expand the 'Position' heading.
- Change the x-coordinate to -500.



- Now first click on the 'Cut' label in the tree view, and then 'ctrl+click' on the 'Cube001' label in the tree view.
- Click on the 'Make a cut of two shapes' icon.
- A new label appears in the tree view 'Cut001'.



A decorative graphic on the left side of the slide, consisting of three overlapping circular frames. The top frame shows a smiling woman talking on a mobile phone. The middle frame shows two large industrial cooling towers. The bottom frame shows two workers in blue uniforms and hard hats working on a large metal component. The bottom-most frame shows a red industrial structure with a complex, branching, root-like design.

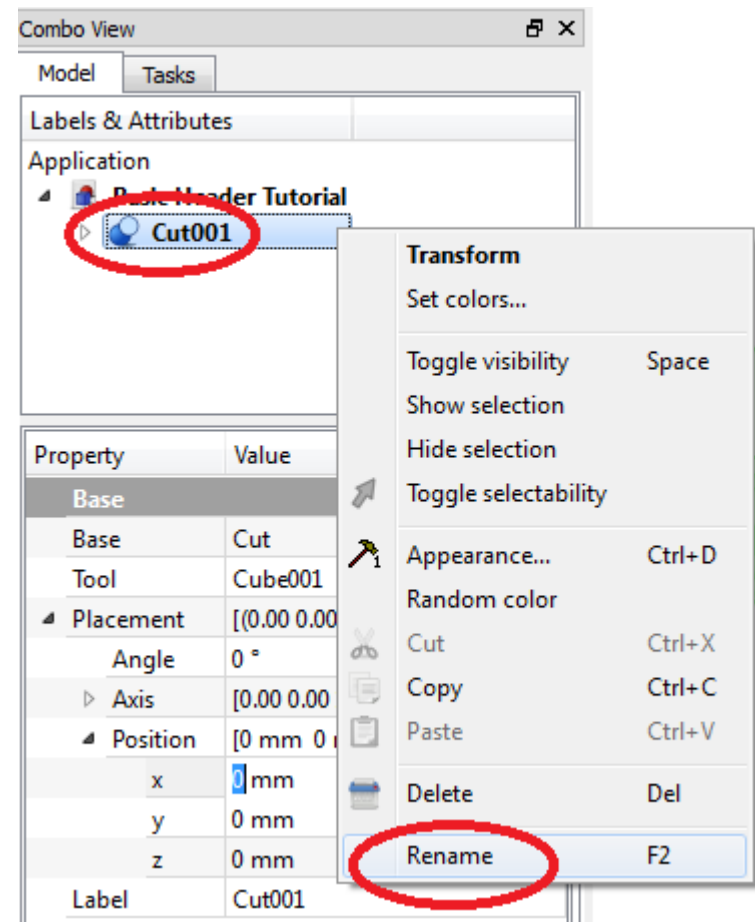
# FEM

Setting up, solving and displaying results for the header.



## Create new analysis with mesh

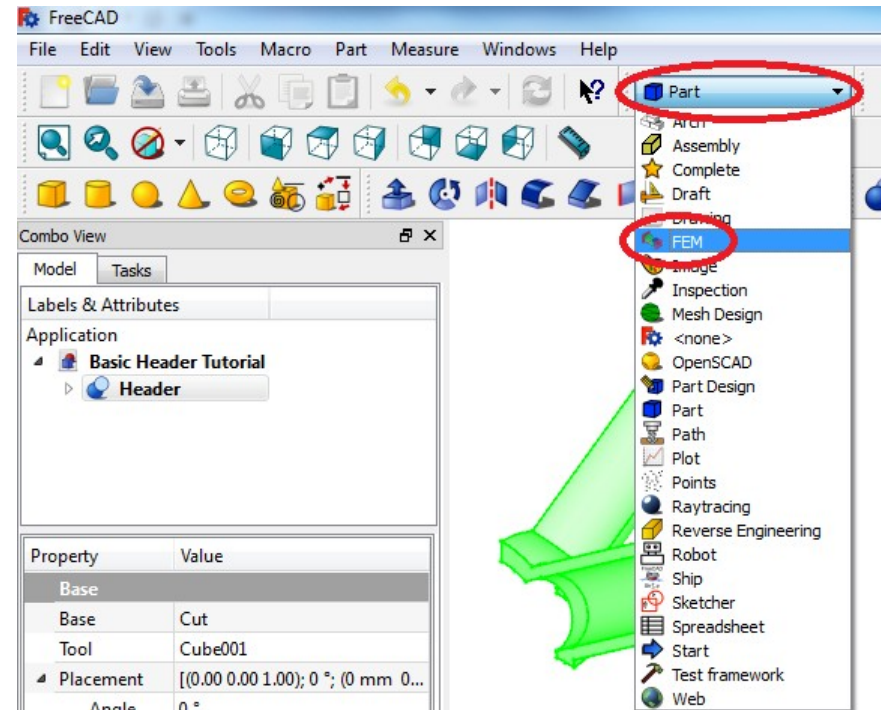
- Now that the CAD model is complete, the Finite Element Analysis (FEA) need to be setup
- First we need to rename the part, since its current name as displayed in the tree view is 'Cut001'
- In the tree view, right click on the label 'Cut001' and click 'Rename'.
- Type in 'Header' and press 'Enter' on the keyboard.
- The name should now be displayed as 'Header'.




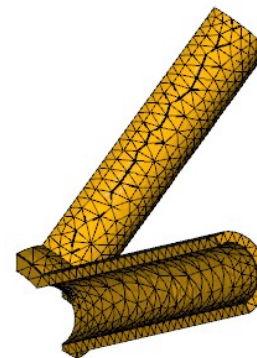
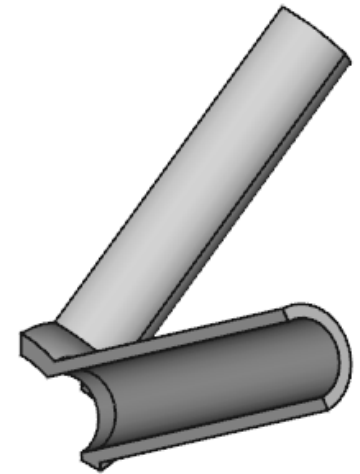
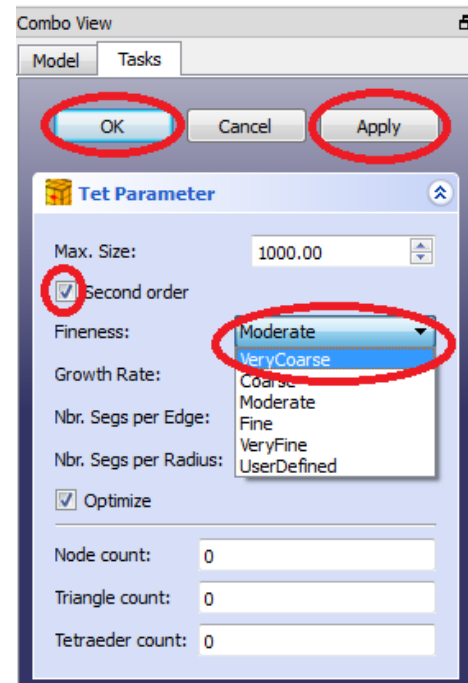


# FEM Model – New analysis


- Click on the workbench dropdown menu in the taskbar and select the 'FEM' workbench.
- Select the part by clicking on the label 'Header' in the tree view. When the part is selected, it will display green in the display window.

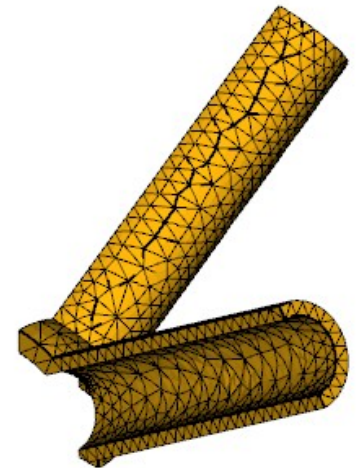
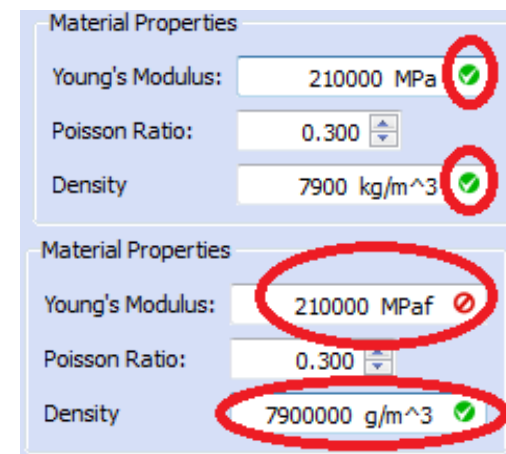
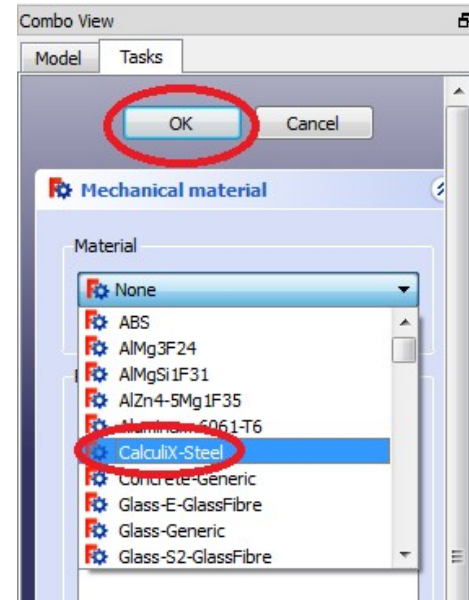


- Click on the 'Create New Mechanical Analysis' icon  on the taskbar. Since the part was selected before clicking on the new analysis, the meshing window will automatically appear.
- Ensure the tick box next to 'Second order' is selected.
- Click on the dropdown menu next to 'Fineness', and select 'VeryCoarse'. This is the fineness of the mesh. Remember to always start with a coarser mesh since the solution time is shorter. When the model solves correctly, the mesh fineness can be adjusted.




## Adding a material

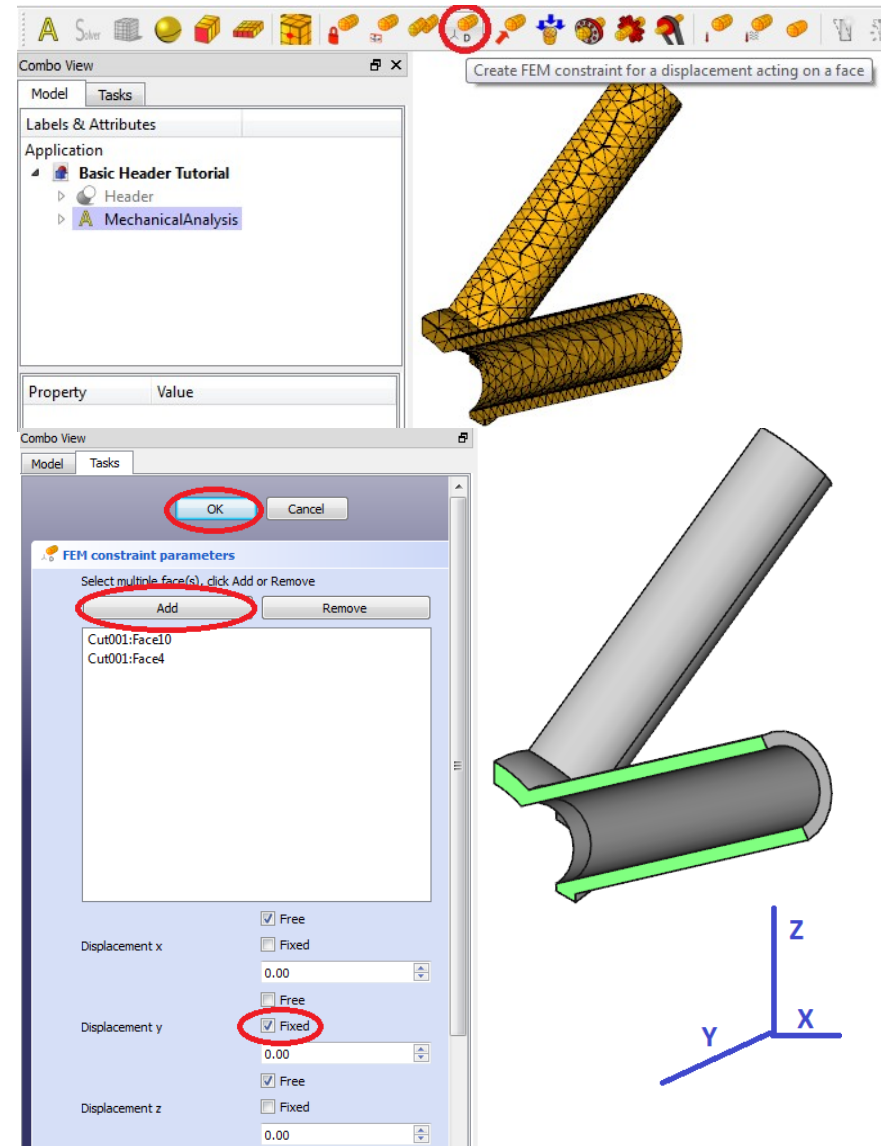
- Click on the 'Add Material' icon  on the taskbar. A dialogue box will appear.
- Click on the dropdown menu under 'Material', and select the generic material called 'CalculiX-Steel'.
- Ensure the material properties are correct, including the units. If a green tickmark is displayed next to the unit, then the unit is acceptable. If a red crossed-out circle is displayed, then the units are wrong or inconsistent.
- Click on 'OK' to close the dialogue box.



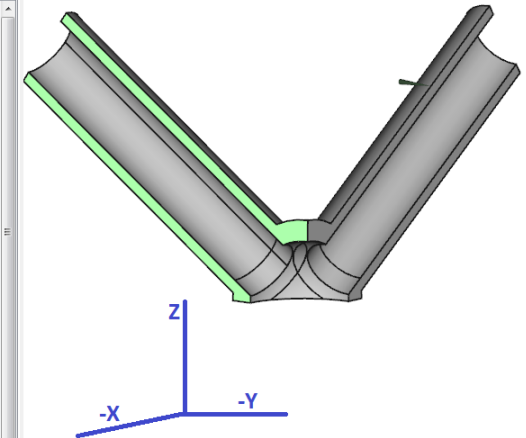
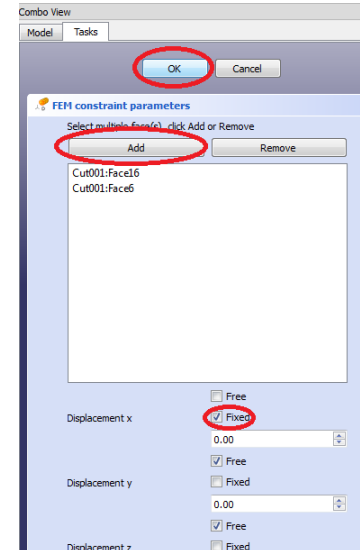
## Compensating for symmetry

- In order to prevent rigid body motion, the model need to be constrained in all three directions. This will be done with the constraints added to compensate for symmetry.
- Since the two branches were cut in half, the cross-sectional area need to be constrained to remain fixed in the direction normal to the surface, since this is what will happen if an internal pressure was applied to the entire pipe.
- We will use the constraint called 'Displacement Constraint'.

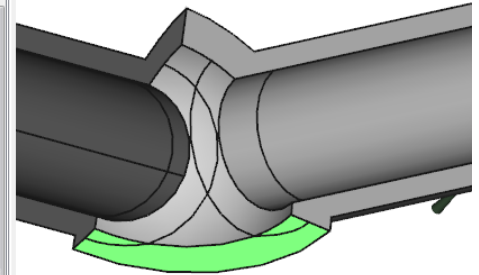
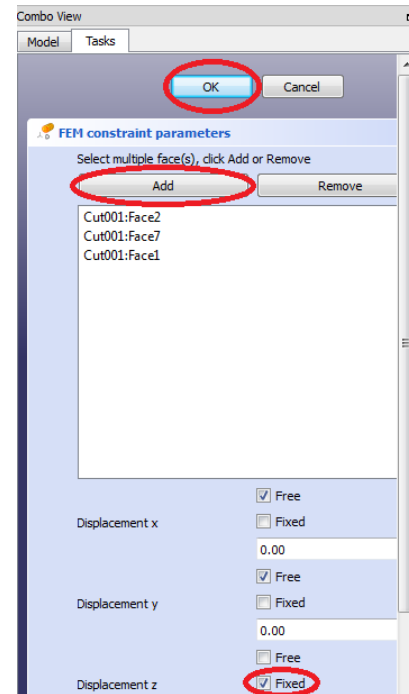
- Click on the 'Add Displacement Constraint'  on the taskbar.
- Select the cross-sectional surface normal to the y-axis by using 'ctrl+click', and click on the 'Add' button. The surfaces will appear on the list.
- Click the 'Fixed' tick box next to the 'Displacement y' heading to fix the displacement in the y-direction.
- Click on 'OK' to close the dialogue box.



- A symmetry constraint need to be applied to the other branch as well.
- Click on the 'Add Displacement Constraint' icon in the taskbar.
- Rotate the model in order to view the cross-sectional area of the branch normal to the x-axis.
- Select the cross-sectional surfaces of the branch normal to the x-axis by using 'ctrl+click'. Click 'Add' to add the surfaces to the list.
- Select the 'Fixed' check box next to 'Displacement x' to fix the displacement of the selected surfaces in the x-direction. Click 'OK' to close the dialogue box.




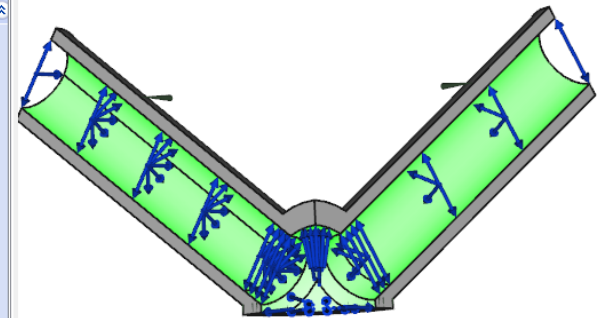
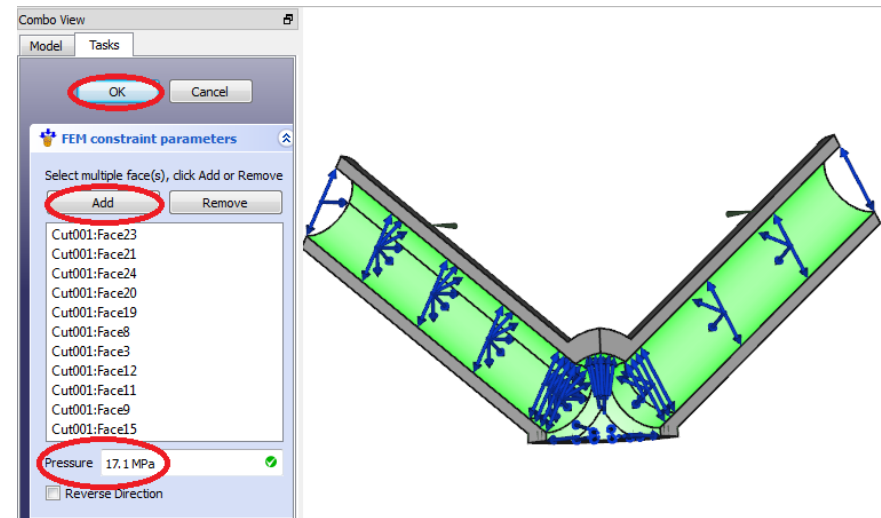
- The final symmetry constraint will be added to the bottom surfaces of the model that is normal to the z-axis.
- Click on the 'Add Displacement Constraint' icon.
- Rotate the model to display the bottom of the model. Select the bottom areas normal to the z-axis.
- Select the surfaces by using 'ctrl+click'. Click 'Add' to add the selected surfaces to the list.
- Select the 'Fixed' check box next to 'Displacement z' to fix the displacement in the z-direction.
- Click 'OK' to close the dialogue.





## Adding internal pressure

- The internal pressure due to the steam need to be added. Click the 'Add Pressure Constraint' icon  on the taskbar. A dialogue box opens.
- Rotate the model to display all the internal surfaces, and select it by using 'ctrl+click'. Click on 'Add' to add the selected surfaces to the list. If a lot of surfaces need to be selected, do it a couple of surfaces at a time.
- Change the pressure to 17.1 MPa.
- Click 'OK' to close the dialogue box.



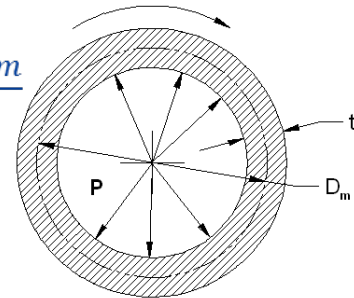


## Adding longitudinal stress

- The internal pressure that was just added represents the hoop stress component, but the longitudinal stress component still need to be presented as well. A force applied the surface at the end of the pipe can be added for this.
- The calculation needed for a full pipe is shown on the right.
- $P$  = Internal pressure of the pipe
- $D_m$  = mean diameter (internal diameter plus thickness)
- $t$  = Wall thickness of pipe


## Longitudinal stress for a full pipe

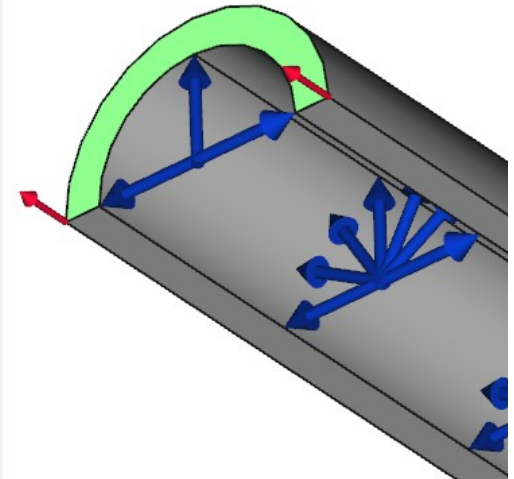
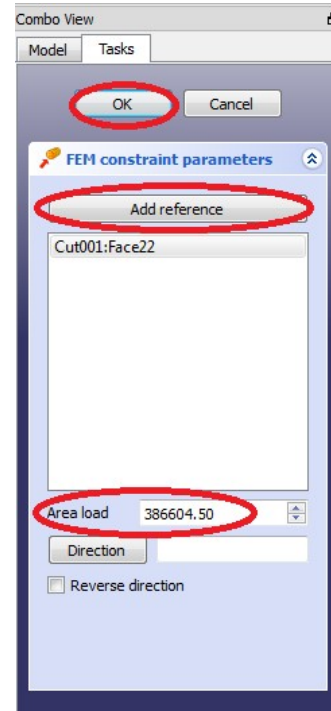
- $\sigma_{long} = \frac{P \times D_m}{4 \times t}$
- $\sigma_{long} = \frac{17.1 \text{ MPa} \times (205 + 35) \text{ mm}}{4 \times 35 \text{ mm}}$
- $\sigma_{long} = 29.3 \text{ MPa}$
- $F = \sigma_{long} \times A_{cross}$
- $F = 29.3 \text{ MPa} \times \frac{\pi}{4} (275 \text{ mm}^2 - 205 \text{ mm}^2)$
- $F = 773\,209 \text{ N}$



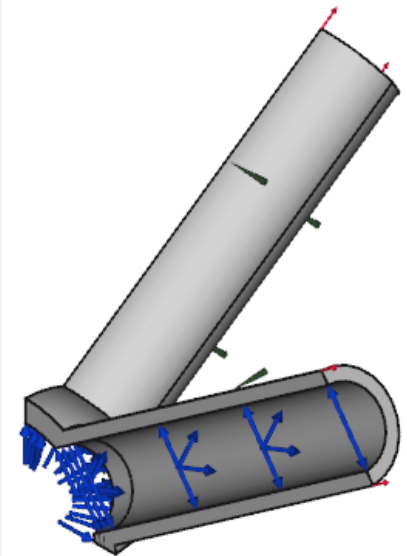
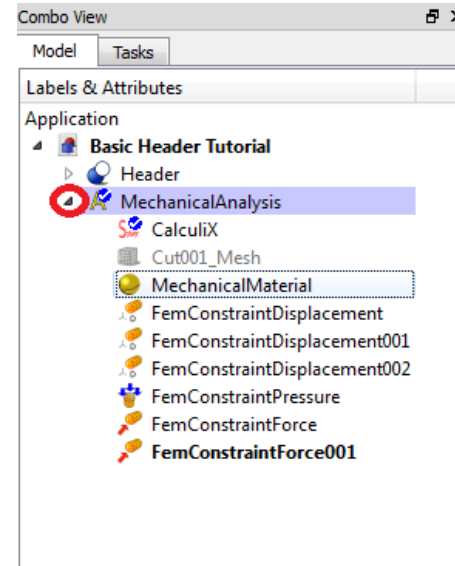
- Since we model only half a pipe, we only apply the force to half the cross-sectional area. Due to this, the force need to be multiplied by half:

- $F = \sigma_{long} \times (0.5 \times A_{cross})$
- $F_{half} = 0.5 \times F = 0.5 \times 773\,209 \text{ N} = 386\,604.5 \text{ N}$

- Click on the 'Add a Force' constraint icon  on the taskbar. A dialogue box opens.
- Click 'Add reference' and then click on the cross-sectional area at the tip of one of the pipes. Since the direction will differ for the other pipe, we need to add a separate force constraint for the other pipe.
- Change the Load to the calculated load needed for the longitudinal stress (386 604.5N). If arrows in the display window points in the opposite direction, click on the 'reverse direction' box to reverse the direction. Click 'OK' to close the dialogue.

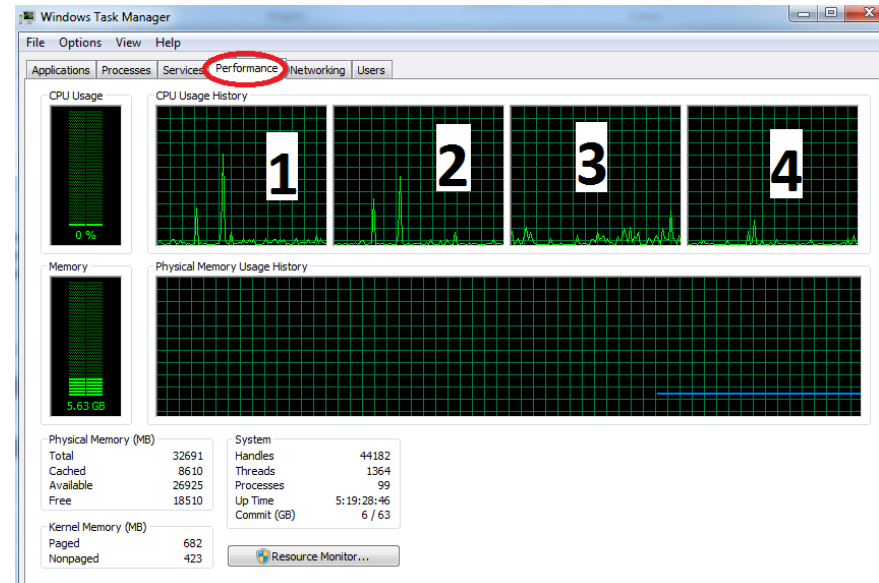


- Similarly Add another force constraint for the other branch as well, using the same value for the applied force.
- In order to view all the constraints added to the model, click on the white arrow next to the 'MechanicalAnalysis' label in the tree view. This will expand all the features and constraints relevant to the analysis.
- It should look similar to the image displayed on the right.

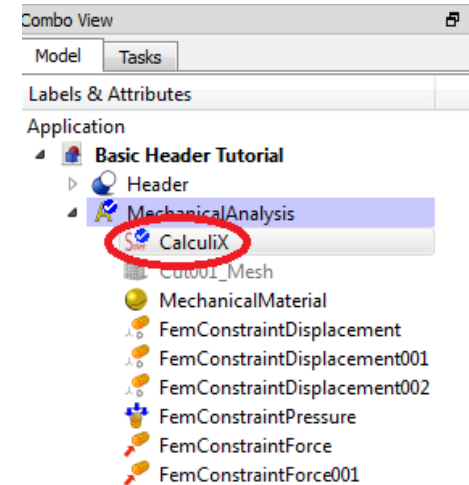


## Setting the number of CPU's

- Before the analysis is run, set the number of CPU's the solver can use. This will optimise the performance and decrease the solution time.
- Determine the amount of CPU's on your computer by pressing 'ctrl+alt+del' and then by selecting 'Start Task Manager'. The Windows Task Manager window opens.
- Select the performance tab at the top and count the number of boxes under 'CPU Usage History'. This is the amount of CPU's your computer has.



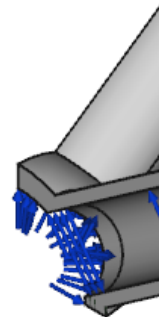
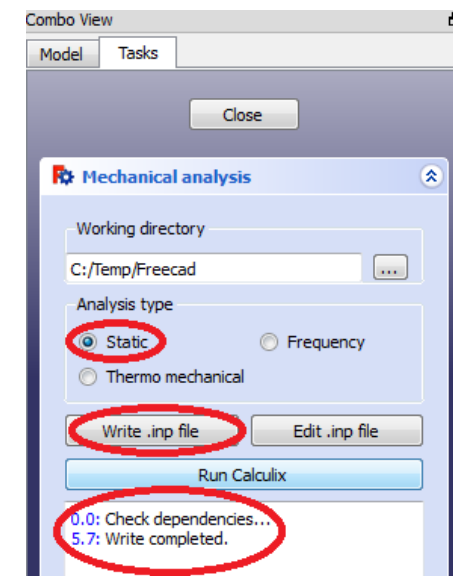
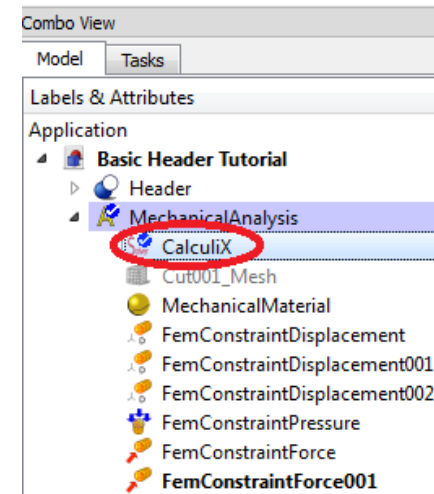
- After determining the number of CPU's on your computer, expand the 'MechanicalAnalysis' label in the tree view by clicking on the arrow next to it.
- Click once on the 'CalculiX' label. This opens the properties for the solver.
- Enter the number of your CPU's minus 1 in the box next to 'Num Cpus'. The reason to leave one CPU open is to give the other system processes one CPU to use.



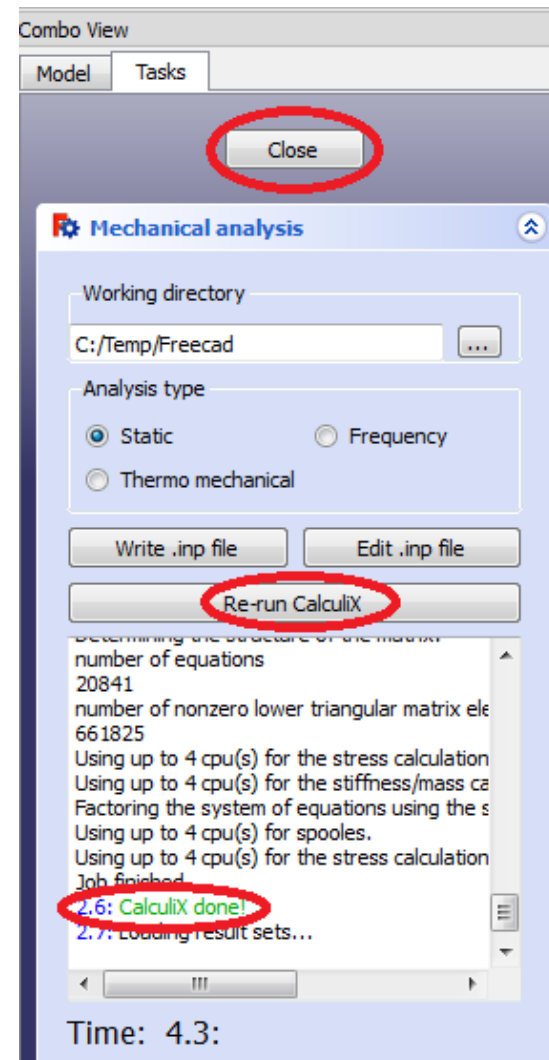
Property	Value
Base	
Label	CalculiX
Solver Type	FemSolverCalculix
Fem	
Analysis Type	static
Eigenmode ...	1 000 000.00
Eigenmode ...	0.00
End Time	1.00
Initial Time ...	0.01
Matrix Solve...	default
Maxiterations	2000
Non-Linear ...	false
Num Cpus	3
Number Of ...	10
Steady State	true
Working Dir	C:/Temp/Freecad

## Running the analysis

- All the constraints have been added and the problem can now be run.
- In the tree view, expand the label 'MechanicalAnalysis' by clicking on the white arrow next to it.
- Double-click on the 'CalculiX' label. This will open a dialogue box for the CalculiX solver.
- Click on the radio button next to 'Static'.
- Click on 'Write .inp file'. Wait until it's finished.

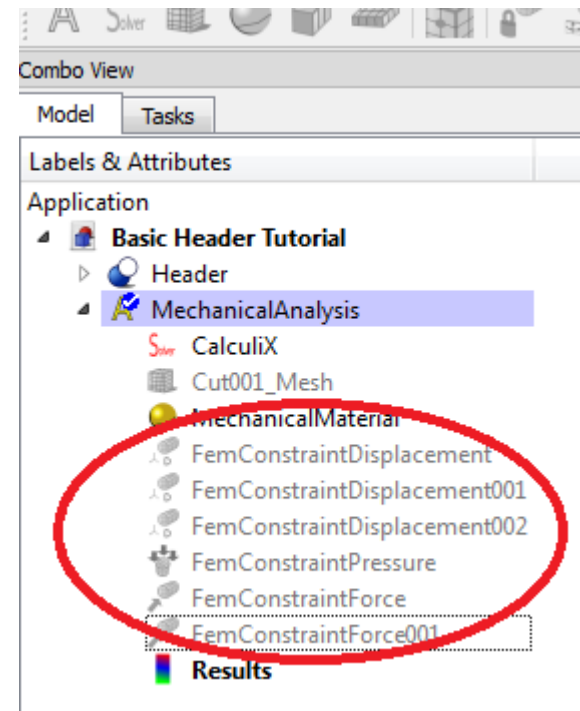


- After the text 'Write completed' is displayed underneath the 'Run CalculiX' button, the 'Run CalculiX' button can be clicked to run calculix.
- Wait for the solver to finish by displaying the text 'Calculix done!' in green in the text box underneath the 'Run CalculiX' button.
- Click 'Close' to close the dialogue.



## Display results using 'Results' icon

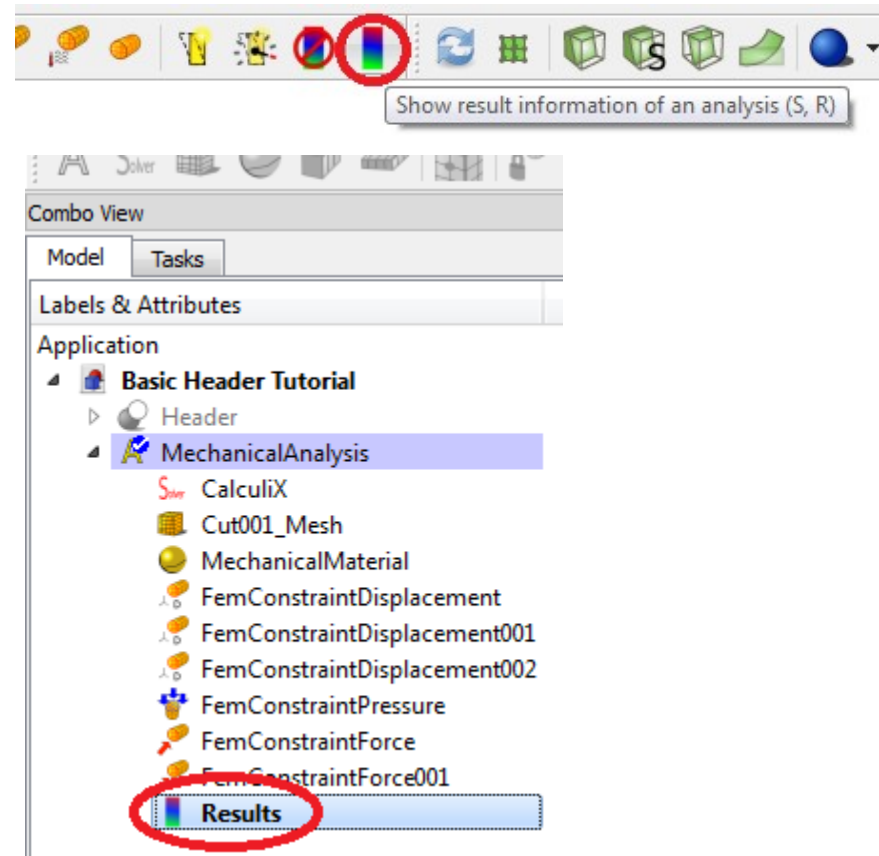
- After the 'CalculiX' solver have been run and the dialogue box have been closed, the results can now be viewed.
- But in order to be able to view the results properly, the constraints that are visible in the in the display window need to be hidden.
- In the tree view, click once on the first constraint and press spacebar to hide the constraint. The label for the constraint will turn grey. Pressing spacebar again will make the constraint visible again. Repeat this until all constraints are hidden.



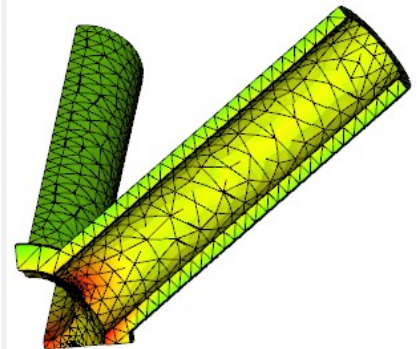
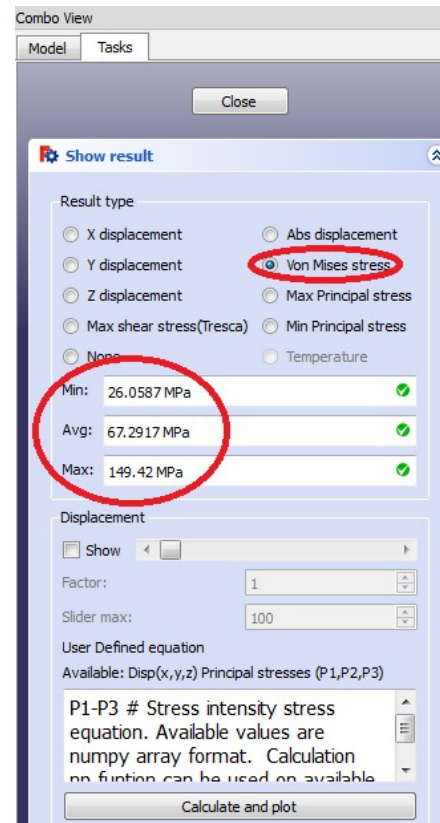


## Display results using 'Results' icon

- To view the results, click on the 'Display Results' icon on the taskbar or double-click the 'Results' label in the tree view to open the results dialogue box.

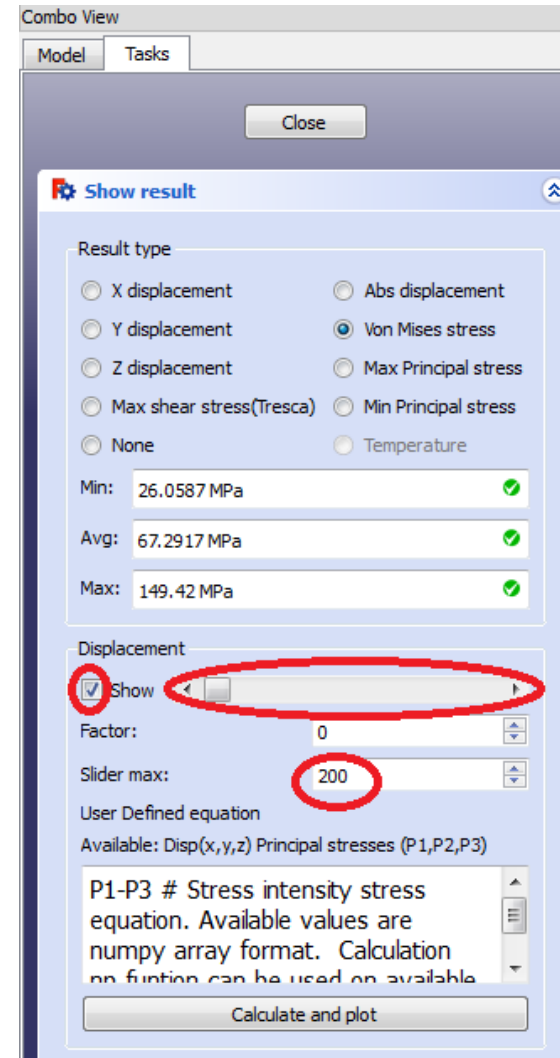


- The mesh will automatically appear on the model in the display window. When you click on the radio button of one of the results, for instance 'Von Mises stress', a color gradient of the results will appear on the model and its minimum, maximum and average values will be displayed in the results block.



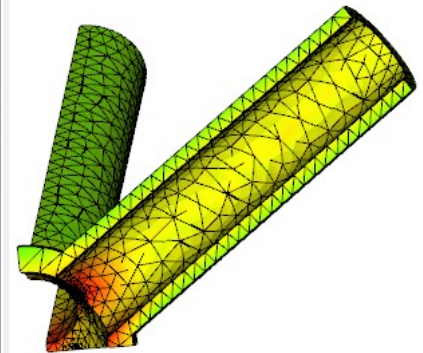
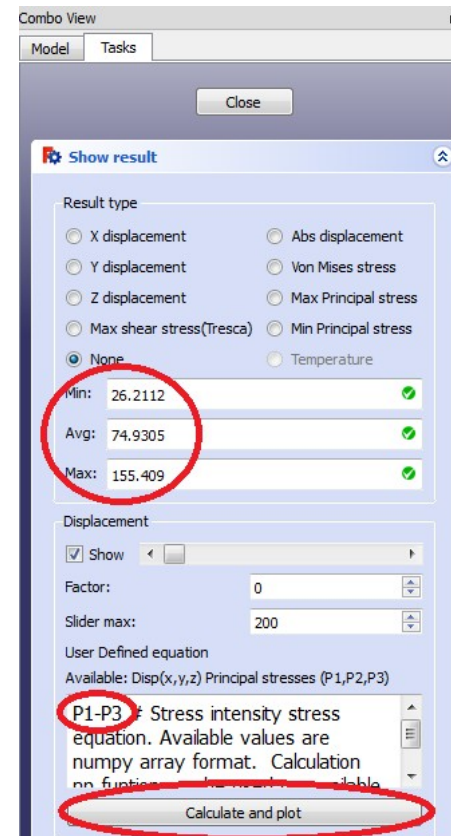
## Displacement using slider

- The displacement of the model can also be shown by magnifying its effect in the display window.
- Under 'Displacement', select the tick box next to 'Show', and slide the sidebar to see an exaggerated displacement of the part.
- If the displacement is not exaggerated enough, the maximum value of the slider can be adjusted in the box next to 'Slider max' to increase the exaggeration of the displayed displacement.



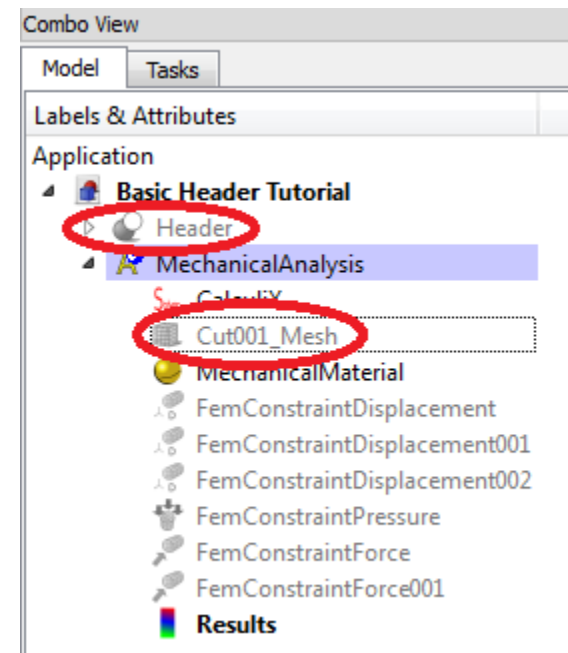
## Display user-defined results


- If you want to display different values (like stress intensity, which is  $P_1 - P_3$  of your principal stresses), you can define it in the text box at the bottom and click 'Calculate and plot' to show the colour gradient on the sketch, and its maximum, minimum and average values will display in the relevant boxes.
- Click on 'Close' to close the results dialogue.

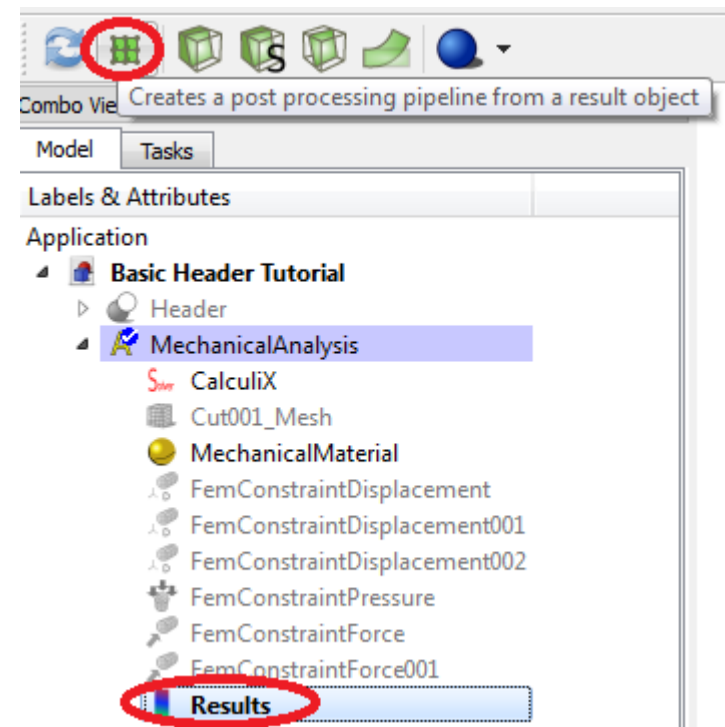


## Displaying basic results using VTK

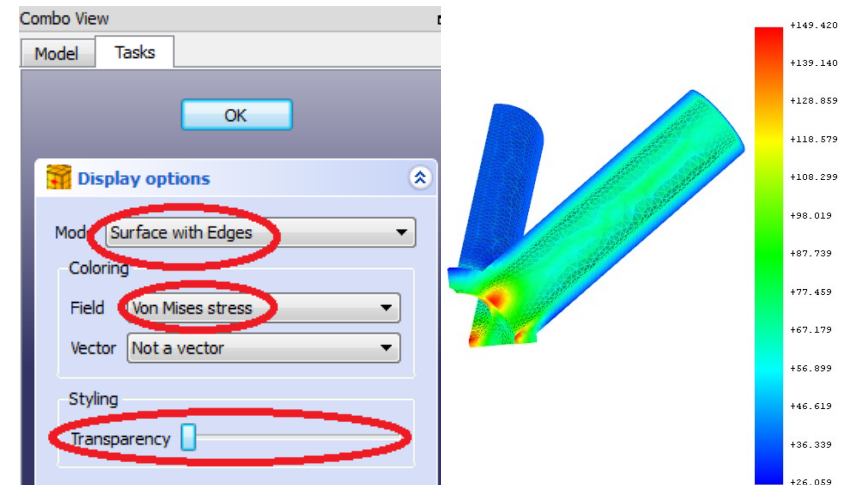
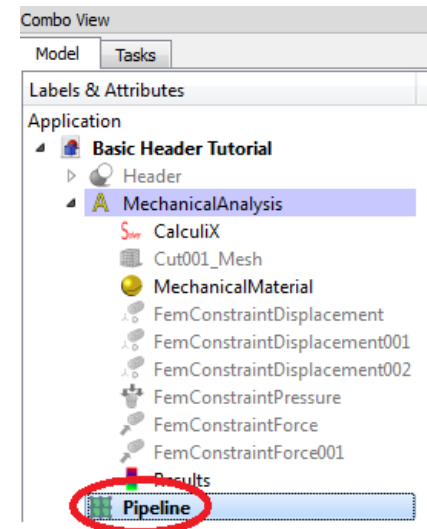
- Before the results using the VTK post-processor can be viewed, the mesh and/or model displayed in the display window need to be hidden.
- In the tree view, if the 'Header' label representing the CAD model is not greyed out, click on it once and press spacebar.
- In the tree view, if the label 'Cut001\_Mesh' is not greyed out, click on it once and press spacebar to hide it.
- The results window should now be empty and the model and the mesh labels should be grey.



- In order to display the results via VTK post-processing, the post-processor needs to be linked to a result.
- In the tree view, click once on the 'Result' label and click on the 'Create Post-Processing Pipeline' icon  in the taskbar. A new label will appear in the tree view for the pipeline.



- Double-click the 'Pipeline' label in the tree view to open the display options for the VTK post-processor pipeline.
- Click on the dropdown menu next to 'Mode' and select 'Surface with edges'. A grey representation of your model will appear after a short while.
- Under 'Coloring', next to 'Field', select 'Von Mises' stress. The grey model becomes coloured and the gradient scale presents the values for the Von Mises stress.
- Experiment with different modes and field colourings and adjust the transparency as required.





Thank you